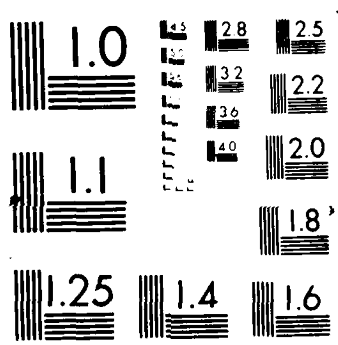
AD-A185 940 USER MANUAL FOR ATILA A FINITE-ELEMENT CODE FOR MODELING PIEZOELECTRIC TRANSDUCERS(U) NAVAL POSTGRADUATE SCHOOL MONTEREY CA J DECARPIGNY ET AL. 1/2

UNCLASSIFIED SEP 87 NPS61-87-007 F/G 17/1 NL


1.0
2.8
2.5
3.2
2.2
3.6
4.0
2.0
1.1
1.8
1.25
1.4
1.6

NPS61-87-007

NAVAL POSTGRADUATE SCHOOL

Monterey, California

USER MANUAL FOR ATILA,
A FINITE-ELEMENT CODE FOR MODELING
PIEZOELECTRIC TRANSDUCERS

Jean-Noel Decarpigny
and
Jean-Claude Debus

August 1987

Prepared for:
Naval Sea Systems Command
Washington, DC

UNCLASSIFIED

SECURITY CLASSIFICATION OF THIS PAGE

REPORT DOCUMENTATION PAGE

A185 948

1a. REPORT SECURITY CLASSIFICATION UNCLASSIFIED	1b RESTRICTIVE MARKINGS
2a. SECURITY CLASSIFICATION AUTHORITY	3. DISTRIBUTION / AVAILABILITY OF REPORT Approved for public release, unlimited distribution
2b. DECLASSIFICATION / DOWNGRADING SCHEDULE	
4. PERFORMING ORGANIZATION REPORT NUMBER(S) Naval Postgraduate School 61-87-007	5. MONITORING ORGANIZATION REPORT NUMBER(S)
6a. NAME OF PERFORMING ORGANIZATION Naval Postgraduate School Department of Physics — 6b. OFFICE SYMBOL (If applicable)	7a. NAME OF MONITORING ORGANIZATION
6c. ADDRESS (City, State, and ZIP Code) Monterey, CA 93943-5101	7b. ADDRESS (City, State, and ZIP Code)
8a. NAME OF FUNDING / SPONSORING ORGANIZATION Naval Sea Systems Command Naval Research Laboratory USRD — 8b. OFFICE SYMBOL (If applicable)	9. PROCUREMENT INSTRUMENT IDENTIFICATION NUMBER N0002487P001388
8c. ADDRESS (City, State, and ZIP Code) ATT R. W. Timme P. O. Box 8337 Orlando, FL 32856-8337	10. SOURCE OF FUNDING NUMBERS PROGRAM ELEMENT NO. / PROJECT NO. / TASK NO / WORK UNIT ACCESSION NO.

11. TITLE (Include Security Classification)
USER MANUAL FOR ATILA, A FINITE-ELEMENT CODE FOR MODELING PIEZOELECTRIC TRANSDUCERS

12. PERSONAL AUTHOR(S)
Jean-Noel Decarpigny and Jean-Claude Debus

13a. TYPE OF REPORT	13b. TIME COVERED	14. DATE OF REPORT (Year, Month, Day)	15 PAGE COUNT
Interim	FROM July 87 TO Aug 87	87/09	

16. SUPPLEMENTARY NOTATION

17. COSATI CODES			18. SUBJECT TERMS (Continue on reverse if necessary and identify by block number)
FIELD	GROUP	SUB-GROUP	ATILA, Modeling, Finite-Element, Piezoelectric Transducer

19. ABSTRACT (Continue on reverse if necessary and identify by block number)

This manual for the user of the finite-element code ATILA provides instruction for entering information and running the code on a VAX computer. The manual does not include the code.

A-1

20 DISTRIBUTION / AVAILABILITY OF ABSTRACT ☒ UNCLASSIFIED/UNLIMITED ☐ SAME AS RPT ☐ DTIC USERS	21 ABSTRACT SECURITY CLASSIFICATION UNCLASSIFIED
22a. NAME OF RESPONSIBLE INDIVIDUAL O. B. Wilson — 22b TELEPHONE (Include Area Code) 408/646-2894	22c OFFICE SYMBOL 61W1

DD FORM 1473, 84 MAR — 83 APR edition may be used until exhausted. All other editions are obsolete

SECURITY CLASSIFICATION OF THIS PAGE

U.S. Government Printing Office: 1986—606-24.

UNCLASSIFIED

```
  AAAAAAAA     TTTTTTTTTT    II    LL            AAAAAAAA
AA        AA       TT        II    LL          AA        AA
AA        AA       TT        II    LL          AA        AA
AAAAAAAAAA         TT        II    LL          AAAAAAAAAA
AA        AA       TT        II    LL          AA        AA
AA        AA       TT        II    LL          AA        AA
AA        AA       TT        II    LLLLLLL     AA        AA
```

FINITE ELEMENT MODELING

OF

PIEZOELECTRIC TRANSDUCERS

AUGUST 1987

USER MANUAL

THE FINITE ELEMENT CODE **ATILA** HAS BEEN DESIGNED BY THE ACOUSTICS LABORATORY OF THE "**INSTITUT SUPERIEUR D'ELECTRONIQUE DU NORD**" (ISEN. LILLE. FRANCE) FOR THE "**GROUPE D'ETUDES ET DE RECHERCHE EN DETECTION SOUS-MARINE**" (GERDSM. DCAN. FRENCH NAVY. TOULON). THE SOFTWARE ENGINEERING IS PROVIDED BY THE **SINAPTEC** COMPANY (LILLE. FRANCE).

CONTENTS

ACKNOWLEDGEMENTS

The authors wish to thank the participants in the first ATILA work shop held in the USA for their contributions to the refinement of this user manual. Although this helped very much in making the manual and the ATILA program more understandable to English speakers, many will still recognize the French accent.

It has been suggested that a second workshop be held, most likely at the Naval Postgraduate School, in the summer of 1988, after users will have had opportunities to compare problem solutions obtained using ATILA with those using other codes. In the meantime, the authors would be appreciative if users of ATILA provide comments and suggestions for its improvement.

The authors also gratefully acknowledge the support provided by the Naval Sea Systems Command's Sonar Transducer Reliability Improvement Program, through the Naval Research Laboratory, Underwater Sound Reference Detachment (R. Timme and C. Ruggiero), and the Naval Sea Systems Command, PMS-390-E58 (C. Allen), which made possible the translation of the program and manual from French to English and its availability to the U. S. Navy.

```
 AAAAAAAA     TTTTTTTTTT     II   LL          AAAAAAAA
AA      AA        TT         II   LL         AA      AA
AA      AA        TT         II   LL         AA      AA
AAAAAAAAAA        TT         II   LL         AAAAAAAAAA
AA      AA        TT         II   LL         AA      AA
AA      AA        TT         II   LL         AA      AA
AA      AA        TT         II   LLLLLLL    AA      AA
```

FINITE ELEMENT MODELING

OF

PIEZOELECTRIC TRANSDUCERS

AUGUST 1987

CHAPTER 1
GENERAL INFORMATION

THE FINITE ELEMENT CODE **ATILA** HAS BEEN DESIGNED BY THE ACOUSTICS LABORATORY OF THE **"INSTITUT SUPERIEUR D'ELECTRONIQUE DU NORD"** (ISEN. LILLE. FRANCE) FOR THE **"GROUPE D'ETUDES ET DE RECHERCHE EN DETECTION SOUS-MARINE"** (GERDSM. DCAN. FRENCH NAVY. TOULON). THE SOFTWARE ENGINEERING IS PROVIDED BY THE **SINAPTEC** COMPANY (LILLE. FRANCE).

1.1 INTRODUCTION

The finite element code **ATILA** has been specifically developed to aid the design of piezoelectric devices, mainly for sonar applications. Thus, it is able to perform the modal analysis of both axisymmetrical and fully three-dimensional piezoelectric transducers. It can also provide their harmonic response under radiating conditions: nearfield and farfield pressure, transmitting voltage response, directivity pattern, electrical impedance, as well as displacement field, nodal plane positions, stress field and various stress criteria... Its accuracy and its ability to describe the physical behavior of various transducers (Tonpilz transducers, double headmass symmetrical length expanders, free flooded rings, flextensional transducers, bender bars, cylindrical and trilaminar hydrophones...) have been checked by modelling more than twenty different structures and comparing numerical and experimental results. These tests were all successful, provided that sufficient care was previously given to the physical meanings of the models. Most of them, as well as the general modelling rules, have been described in reports, papers and communications, which are partially listed in the next section.

ATILA has been designed for the " **Groupe d'Etudes et de Recherche en Detection Sous-Marine, G.E.R.D.S.M.**" from the "**Direction des Constructions et Armes Navales, D.C.A.N.**", in **TOULON (FRANCE)**, by the Acoustics Laboratory of the "**Institut Superieur d'Electronique du Nord, I.S.E.N.**", in **LILLE (FRANCE)**. It has first been a research tool, for specific optimisation purposes, but, due to the request for its use by a large number of people, it has then been made more user friendly and documented, while a specific effort has been carried out to improve its versatility as well as to provide more efficient graphic display abilities. At the present time, **ATILA** has been run on various IBM, VAX and UNIVAC mainframes, as well as a FPS array processor and VAX and MASSCOMP workstations. Nevertheless, **ATILA** is not a "black box" and users are assumed to have a firm background in finite element analysis.

ATILA has been developed over the past 8 years by a lot of people, but mainly by K. ANIFRANI, R. BOSSUT, J.L. CARTON, J.C. DEBUS, J.N. DECARPIGNY, B. DUBUS, R. EDDE, B. HAMONIC, D. MOREL, P. TIERCE at I.S.E.N., and B. TOCQUET and D. BOUCHER at **G.E.R.D.S.M.**. Current developments, including new algorithms, new material description, material loss description and finite element/integral equations coupling are proceeding at I.S.E.N., while software engineering is now provided by the SINAPTEC company, **LILLE (FRANCE)**.

Information and help about **ATILA** can be obtained, under specific conditions, from:

. **J.N. DECARPIGNY**
Acoustics Laboratory
Institut Superieur d'Electronique du Nord
41 boulevard Vauban
59046 LILLE Cedex. FRANCE
Tel: (33).20.30.62.20 Telex: GEFIRN 120.369

. **P. TIERCE**
SINAPTEC Company
41 boulevard Vauban
59800 LILLE. FRANCE
Tel: (33).20.57.80.04 Telex: GEFIRN 120.369

Following this section and the reference list, the reader will find the description of the successive stages of an **ATILA** job and their connection to the manual organisation. Then, detailed information is collected in five chapters.

1.2 REFERENCES

R. BOSSUT, J.N. DECARPIGNY, "An improvment of the finite radiating element formulation. Application to the modelling of a free flooded transducer", communication M4, 106th ASA meeting, San Diego (1983).

J.N. DECARPIGNY, J.C. DEBUS, P.TIERCE, B. TOCQUET, D. BOUCHER, "Application of the finite element method to the characterisation of piezoelectric ceramics", communication WW6, 106th ASA meeting, San Diego (1983).

B. TOCQUET, D. BOUCHER, P. TIERCE, J.C. DEBUS, "Computation of a piezoelectric deep submergence transducer by three dimensional finite element method", communication WW7, 106th ASA meeting, San Diego (1983).

J.C. DEBUS, J.N. DECARPIGNY, P. TIERCE,"Modelisation a l'aide de la methode des elements finis de projecteurs piezoelectique des types Tonpilz et Isabelle", G.E.R.D.S.M., Convention C.81.48.826.309, rapport final (1984).

P. TIERCE, J.N. DECARPIGNY, "Modelisation par elements finis d'un projecteur piezoelectrique de type Janus", G.E.R.D.S.M., Convention C.83.48.826.163, commande 004, rapport final (1984).

J.N. DECARPIGNY, "Application de la methode des elements finis a l'etude de transducteurs piezoelectriques", Doctoral Thesis, Lille University (1984).

J.L. CARTON, J.C. DEBUS, J.N. DECARPIGNY, B. HAMONIC, P. TIERCE, "Notice des programmes ATILA, MOSAIQUE et GRAFTEC, versions 1.1", G.E.R.D.S.M., Convention C.82.48.826.398 (1984).

R. BOSSUT, "Modelisation de transducteurs piezoelectriques annulaires immerges par la methode des elements finis", Doctoral Thesis, Valenciennes University (1985).

P. TIERCE, "Modelisation du transducteur Isabelle par la methode des elements finis", Doctoral Thesis, Valenciennes University (1985).

J.N. DECARPIGNY, J.C. DEBUS, B. TOCQUET, D. BOUCHER, "In-air analysis of piezoelectric Tonpilz transducers in a wide frequency band using a mixed finite element-plane wave method", J. Acoust. Soc. Am., 78, 1499 (1985).

D. BOUCHER, B. TOCQUET, P. TIERCE, J.N. DECARPIGNY, "Finite element study of the farfield limit of symmetrical length-expander transducers", communication JJ6, 110th ASA meeting, Nashville (1985).

J.N. DECARPIGNY, J.C. DEBUS, P. TIERCE, B. HAMONIC, B. TOCQUET, D. BOUCHER, "Modelisation a l'aide de la methode des elements finis de transducteurs piezoelectriques rayonnant dans l'eau", Actes du Colloque DRET sur les tendances actuelles en calcul des structures, Ed. PLURALIS, p.359, Bastia (1985).

B. HAMONIC, J.C. DEBUS, J.N. DECARPIGNY, P.TIERCE, "Developpement d'un element de coque mince a symetrie axiale et a courbure constante", G.E.R.D.S.M., Convention C.82.48.826.398, rapport final (1985).

J.N. DECARPIGNY, J.C. DEBUS, "Modelisation par ATILA de Tonpilz bifrequences avec excitation differenciee des ceramiques", G.E.R.D.S.M., TSM C.82.48.826.613.T, rapport final (1985).

P. TIERCE, J.N. DECARPIGNY, "Etude par elements finis du rayonnement d'un transducteur de type JANUS", G.E.R.D.S.M., Convention C.83.48.826.163, commande 007, rapport final (1985).

B. HAMONIC, D. MOREL, "Implantation du code ATILA sur calculateur vectoriel FPS 164", G.E.R.D.S.M., Convention C.83.48.826.163, commande 008, rapport final (1985).

J.N. DECARPIGNY, J.C. DEBUS, O.B. WILSON, "Finite element analysis of the DT276 hydrophone using the ATILA code", Report number 61-86001, Naval Postgraduate School, Monterey (1986).

B. HAMONIC, J.C. DEBUS, J.N. DECARPIGNY, D. BOUCHER, B. TOCQUET, "Application of axisymmetrical thin shell finite elements to the analysis of a radiating flexural shell sonar transducer", International symposium on practical aspects in the computation of shell and spatial structures, Leuven (1986).

B. HAMONIC, R. BOSSUT, J.C. DEBUS, J.N. DECARPIGNY, D. MOREL, P. TIERCE, D. BOUCHER, B. TOCQUET, "Modal analysis of sonar transducers using the finite element method", 11th international seminar on modal analysis, Leuven (1986).

R. BOSSUT, J.N. DECARPIGNY, D. BOUCHER, B. TOCQUET, "Application of damping elements to the modelling of underwater radiating structures", communication X8, 111th ASA meeting, Cleveland (1986).

B. HAMONIC, J.C. DEBUS, J.N. DECARPIGNY, D. BOUCHER, B. TOCQUET, "Analysis of radiating flexural shell sonar transducers using the finite element method", communication L4, 112th ASA meeting, Anaheim (1986).

D. BOUCHER, B. TOCQUET, J.N. DECARPIGNY, J.C. DEBUS, P. TIERCE, "Effect of a voltage shading of the ceramic stack on the bandwidth of a radiating Tonpilz transducer", communication L9, 112th ASA meeting, Anaheim (1986).

B. HAMONIC, J.C. DEBUS, J.N. DECARPIGNY, "Analyse modale de transducteurs flextensionnels a l'aide du code elements finis ATILA", G.E.R.D.S.M., Convention C.83.48.826.163, commande 024, rapport final (1986).

J.C. DEBUS, D. MOREL, J.N. DECARPIGNY, "Incorporation au code elements finis ATILA d'une methode de stockage par colonnes actives", G.E.R.D.S.M., Convention C.83.48.826.163, commande 016, rapport (1986).

J.N. DECARPIGNY, J.C. DEBUS, B. HAMONIC, R. BOSSUT, P. TIERCE, D. MOREL, D. BOUCHER, B. TOCQUET, "Finite element analysis of low frequency sonar transducers", Proceedings of the two-day conference on sonar transducers, Birmingham (1987).

B. STUPFEL, C. GRANGER, J.N. DECARPIGNY, "Finite element modelling of a pressure-gradient hydrophone", Proceedings of the two-day conference on sonar transducers, Birmingham (1987).

B. DUBUS, J.C. DEBUS, J.N. DECARPIGNY, D. MOREL, D. BOUCHER, "Determination of the mechanical limits of piezoelectric transducers using the finite element method", communication MM2, 113th ASA meeting, Indianapolis (1987).

B. HAMONIC, "Application of the finite element method to the design of power piezoelectric sonar transducers", Proceedings of the international workshop on power sonic and ultrasonic transducer design, Ed. Springer Verlag, Lille (1987).

P. TIERCE, J.N. DECARPIGNY, "Design of high power ultrasonic transducers for use in macrosonics", Proceedings of the international workshop on power sonic and ultrasonic transducer design, Ed. Springer Verlag, Lille (1987).

B. HAMONIC, "Contribution a l'etude, par la methode des elements finis, du rayonnement de transducteurs utilisant des vibrations de coques minces", Doctoral Thesis, Lille University (1987).

B. HAMONIC, J.C. DEBUS, J.N. DECARPIGNY, "Modelisation d'un transducteur Olive a l'aide du code elements finis ATILA", G.E.R.D.S.M., Convention C.83.48.826.163, commande 013, rapport final (1987).

B. DUBUS, J.C. DEBUS, J.N. DECARPIGNY, "Application du code elements finis ATILA au calcul des contraintes statiques et dynamiques dans les transducteurs piezoelectriques", G.E.R.D.S.M., Convention C.83.48.826.163, commande 027, rapport final (1987).

K. ANIFRANI, J.C. DEBUS, J.N. DECARPIGNY, "Developpement de la bibliotheque des elements piezoelectriques 2D et 3D du code ATILA", G.E.R.D.S.M., Convention C.83.48.826.163, commande 021, rapport final (1987).

B. STUPFEL, A. LAVIE, J.N. DECARPIGNY, "Implantation dans le code ATILA du couplage elements finis et equations integrales", G.E.R.D.S.M., Convention C.83.48.826.163, commande 005, rapport final (1987).

D. MOREL, "Post-traitements tridimensionnels, isovaleurs et animations", Colloque sur l'apport des techniques graphiques au calcul des structures, Ed. IPSI, Paris (1987).

1.3 GENERAL ORGANISATION OF AN ATILA JOB

Running an **ATILA** job requires different well defined stages. This section describes briefly their organisation and their connections to the various following chapters of this manual.

1.3.1. THE MODEL DEFINITION.

Being given the physical problem, the user has to define the **type of analysis** which is required: computation of the static deformation of an elastic stucture under lumped mechanical load, modal analysis of a piezoelectric stack with short circuit electrical condition, computation of the transmitting voltage response of a transducer... This type of analysis has to be related to **solving algorithms** which are listed and described in **chapter 2**. New solvers are currently under test or development in I.S.E.N. Acoustics laboratory and detailed information can be provided upon request. The user has also to determine the **type(s) of elements** which is (are) required: three-dimensional elements, axisymmetrical, plane stress or plane strain elements, plate or shell elements relying on specific approximations... The **ATILA element library** is described in **chapter 4**.

At this stage, a **first estimation of the computation size** can generally be made, using a gross determination of the node and degree of freedom numbers and previous CPU time measurements or classical numerical rules. This estimation is important since it can avoid the generation of unrealistic meshes. At this stage also, the user has to gather the **requested physical parameters**, as described in **chapter 4**, and to ensure their accuracy, which very definitely determines the accuracy of the result. Though not directly involved in the data file generation, this stage is fundamental.

1.3.2. THE MESH GENERATION.

Following the previous choice of element type(s), as defined during the first stage, the **mesh generation** implies the splitting of the whole domain into a given number of elements. This is realised by defining nodes and assigning them to corresponding elements. **Node definition** is made by listing the node coordinates in a given order, called the node numbering order. Then, **assignment of the nodes to the corresponding elements** is obtained by listing for each element, in a given order named its **topology**, the nodes which belong to it. The node coordinate description is detailed in **chapter 3**, together with all other entries. The topology description is presented also in **chapter 3**, after the node coordinate description, but more details are available on an element by element basis in **chapter 4**.

During the mesh generation, the user can use the isoparametric character of all the **ATILA** elements, which allows easy curved line modelling and lower density meshes. Nevertheless, a great deal of attention has to be paid to the element geometrical aspect, as well as to the mesh step which has to be related, for example, to the minimal wavelength value of interest.

Mesh generation can be fully carried out by the user. However, in many cases, partial regularity of the mesh allows the use of an **automatic generation** which provides node coordinates and element topologies. Then, the user has only to define a gross splitting, built with **super elements**, and to select their automatic splitting into finite elements. This easier procedure, described in **chapter 6**, relies upon the use of the **MOSAIQUE** code.

1.3.3. THE DATA FILE PREPARATION.

An **ATILA** data file includes **entries, specific data and comments**. It is devoted to the description of the type of analysis, the node coordinates, the element topology, the material properties, the element geometrical properties, which are all **entries**, and the loading and boundary conditions which are **formatted data**. In some cases, it also includes specific mixed finite element/plane wave model data and graphic display data. The whole data file preparation is described in **chapter 3** which has to be carefully read. Specific element information which are required during this preparation can also be found in **chapter 4**. Finally, when the automatic mesh generation code **MOSAIQUE** is used, data file preparation can be, in a large part, automatically obtained, as explained in **chapter 6**.

A detailed check of the data file can be processed by running a specific progam, named **PGRAPH**, which produces a complete listing of the various data, including the generated degree of freedom numbering which reflects exactly the prescribed boundary conditions, as well as different kinds of graphic displays (total or partial mesh, with or without element, node or degree of freedom numberings). The use of **PGRAPH** is described in **chapter 5**.

1.3.4. THE RUNNING OF THE CODE.

Running of the code requires the edition of a **main program** which has to set the array sizes, call successive subroutines to read the data file, compute the elementary stiffness and mass matrices, assemble these matrices into the global stiffness and mass matrices, solve the equations and display the results. This **progam generation is automatic**, given the corresponding data file, and is provided by running the program **PGEN**. **PGEN** will issue successive questions, on the terminal screen, to specify various user's requests, such as array storages for a delayed solving, array storages for a postprocessing, graphic displays... It will provide various files, including the main program FORTRAN source and, in the case of a VAX computer under VMS operating system, a COMMAND file to compile this main program and edit the links.

Running of the code will provide a detailed result listing and, according to the specific requests, graphic displays or stored arrays for restarting another solving or for postprocessing. It has to be noted that **PGEN** is also able to generate specific main programs to restart given runs. Every information which concerns the program running as well as **PGEN** is contained in **chapter 5**.

1.3.5. RESULT USING AND POSTPROCESSING.

As soon as the run is completed, a **LISTING** file is available which contains all the results: eigenfrequencies and eigenvectors, displacement and pressure fields, electrical impedance... If requested, specific arrays are also stored for postprocessing, and mainly the array **PLO** which contains displacement and pressure fields for every loading case, mode number or excitation frequency. Graphic displays of the distorted structure can be easily obtained on various workstations using **PGRAPH**. **PGRAPH** will issue FORTRAN and COMMAND files which rely upon the **GKS** package and have to be executed to provide the display. As mentioned previously, the use of **PGRAPH** is described in **chapter 5**.

1.3.6. SUMMARY.

As a summary, the next page displays a simple flow chart of an **ATILA finite element modelling**. A large part of this flow chart will be reproduced in a more detailed and accurate fashion in **chapter 5**, with reference to the various files and programs.

1. MODEL DEFINITION.

Choice of the type of analysis.
Choice of the elements.
Choice of the solving algorithms.
Computation size estimation.
Gathering of all physical parameters.

2. MESH GENERATION (USER OR MOSAIQUE).

Node definition: **node coordinate list.**
Node appropriation to the elements:
element topologies.
Check of element geometrical aspects.
Check of mesh steps.

3. DATA FILE PREPARATION.

Definition of entries, including **node coordinate list and element topologies.**
Definition of specific data: loads,
boundary conditions, mixed model.
Check of the mesh and data file: **PGRAPH.**

4. RUNNING OF THE CODE (FIRST RUN OR RESTART).

Generation of the main program: **PGEN.**
Specification of user's requests: array
storages, graphic displays...
Run processing.

5. RESULT USING AND POSTPROCESSING.

Reading of the LISTING file.
Graphic display of the distorted shape of
the structure: **PGRAPH.**
Postprocessing using various resulting arrays.

```
 AAAAAAAA    TTTTTTTTTT   II   LL          AAAAAAAA
AA      AA       TT       II   LL         AA      AA
AA      AA       TT       II   LL         AA      AA
AAAAAAAAAA       TT       II   LL         AAAAAAAAAA
AA      AA       TT       II   LL         AA      AA
AA      AA       TT       II   LL         AA      AA
AA      AA       TT       II   LLLLLLL    AA      AA
```

FINITE ELEMENT MODELING

OF

PIEZOELECTRIC TRANSDUCERS

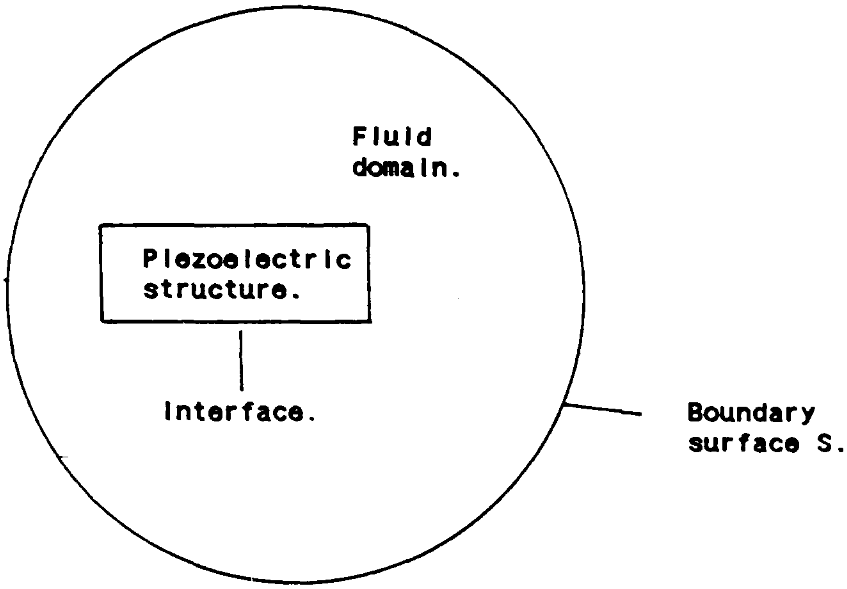
AUGUST 1987

CHAPTER 2
TYPES OF ANALYSIS

THE FINITE ELEMENT CODE **ATILA** HAS BEEN DESIGNED BY THE ACOUSTICS LABORATORY OF THE "**INSTITUT SUPERIEUR D'ELECTRONIQUE DU NORD**" (ISEN. LILLE. FRANCE) FOR THE "**GROUPE D'ETUDES ET DE RECHERCHE EN DETECTION SOUS-MARINE**" (GERDSM. DCAN. FRENCH NAVY. TOULON). THE SOFTWARE ENGINEERING IS PROVIDED BY THE **SINAPTEC** COMPANY (LILLE. FRANCE).

2.1 GENERAL FORMULATION

This section describes the general formulation which has been retained to model radiating piezoelectric transducers using the **ATILA** code and provides a complete list of the available types of analysis. Then, for each type of analysis, the following sections give more information and specific details.



To model a radiating piezoelectric transducer using the **ATILA** code, the finite element mesh must include the structure as well as a part of the fluid domain, as described on the above figure. Thus, the unknown fields are the displacement field $\underline{u}$ in the whole structure, the electrical potential v in the piezoelectric material and the pressure p in the fluid. Then, the equations which are solved are, first, the equation of motion in the structure, second, the Poisson equation in the piezoelectric material and, third, the Helmholtz equation in the fluid. Kinematic and dynamic continuity equations between displacement and pressure fields are prescribed at the interface, due to the variational formulation, and an appropriate damping condition is applied to the spherical external surface S that surrounds the fluid domain.

In matrix form, the complete set of equations associated with this problem is:

$$\begin{vmatrix} [K_{uu}] - w^2[M] & [K_{uv}] & -[L] \\ [K_{uv}]^T & [K_{vv}] & [0]^T \\ -a^2c^2w^2[L]^T & [0] & [H] - w^2[M_1] \end{vmatrix} \begin{vmatrix} \underline{U} \\ \underline{V} \\ \underline{P} \end{vmatrix} = \begin{vmatrix} \underline{F} \\ -\underline{Q} \\ ac^2\underline{F}' \end{vmatrix}$$

where:

- $\underline{U}$, $\underline{F}$, $\underline{V}$, $\underline{Q}$ and $\underline{P}$ are vectors that contain the nodal values of the displacement field, the lumped applied forces, the electrical potential and charges, the pressure field,
- $\underline{F}'$ is a vector that contains the nodal values of the outgoing flux through S,
- $[K_{uu}]$ and $[H]$ are the solid and fluid stiffness matrices,
- $[M]$ and $[M_1]$ are the solid and fluid mass matrices,
- $[K_{uv}]$ and $[K_{vv}]$ result from the assembling of the piezoelectric and dielectric matrices,
- $[L]$ is the fluid-structure interface connectivity matrix,
- $[0]$ is a null matrix,
- a and c are the fluid density and sound speed,
- w is the circular frequency,
- T means transposed.

In fact, internal electrical potential degrees of freedom are associated with null nodal charges and can be easily condensed. Moreover, nodes which belong to hot electrodes have the same electrical potential degree of freedom which can be factored, following simple algebraic operations. Finally, the reference potential can be arbitrarily put equal to zero. Thus, the preceeding matrix equation is automatically transformed by the **ATILA** code into the following form:

$$\begin{vmatrix} [K_{uu}] - w^2[M] & \underline{K}_{uv} & -[L] \\ \underline{K}_{uv}^T & K_{vv} & \underline{0}^T \\ -a^2c^2w^2[L]^T & \underline{0} & [H] - w^2[M_1] \end{vmatrix} \begin{vmatrix} \underline{U} \\ V \\ \underline{P} \end{vmatrix} = \begin{vmatrix} \underline{F} \\ -I/jw \\ ac^2\underline{F}' \end{vmatrix}$$

where elastic, piezoelectric and dielectric terms are modified by the internal electric potential static condensation, $\underline{K}_{uv}$ and $\underline{0}$ are now vectors, K_{vv} is now a scalar, V is the applied electrical potential and I is the electrical current entering into the structure. Then, if a dipolar damping condition is applied on the external boundary surface S, $\underline{F}'$ is given by:

$$\underline{F}' = -(1/ac).(1/R + jw/c).[D]\ \underline{P}$$
$$+(1/ac).\{(1/R - jw/c)/(1 + w^2R^2/c^2)\}.[D']\ \underline{P}$$

where R is the radius of S, [D] and [D'] are obtained by assembling radiating surface finite elements from S, the first term is the **monopolar** contribution associated with a spherical wave impedance and the second term is the **dipolar** contribution.

Starting with this general problem, a lot of different specific analysis can be processed. Those for which **ATILA** provides a solver are the following:

- computation of the displacement field of an elastic structure under a static lumped or distributed load (**STA 1**),
- computation of the displacement field and the electrical potential for a piezoelectric structure under a static lumped or distributed load (**STA 2**),
- modal analysis of an elastic structure, providing the eigenfrequencies and eigenmodes (**MOD 1**),
- modal analysis of a piezoelectric structure, providing the eigenfrequencies and eigenmodes under resonance or antiresonance electrical conditions (**MOD 2**),
- modal analysis of a closed fluid domain, with zero pressure and/or zero flux boundary conditions, providing the eigenfrequencies and eigenmodes (**MOD 3**),
- modal analysis of a closed hydroelastic system, built with two coupled elastic and closed fluid domains, providing the eigenfrequencies and eigenmodes (**MOD 4**),
- modal analysis of an elastic structure flooded in an incompressible fluid (added mass effect only), providing the eigenfrequencies and eigenmodes (**MOD 5**),
- harmonic analysis of an in-air piezoelectric structure, providing its electrical impedance and displacement field for a given excitation frequency (**HAR 1**),
- harmonic analysis of an in-air piezoelectric structure, using the mixed plane wave/finite element method and providing the displacement field for a given excitation frequency (**HAR 2**),
- radiated pressure computation for a given displacement field of a vibrating structure at a given frequency (**HAR 3**),
- harmonic analysis of a radiating piezoelectric transducer, providing the electrical impedance and the displacement and pressure fields in the whole mesh for each given excitation frequency (**HAR 4**),
- harmonic analysis of a radiating piezoelectric transducer, using the mixed plane wave/finite element method and providing the displacement and pressure fields in the whole mesh for each given excitation frequency (**HAR 5**).

2.2 STATIC ANALYSIS OF AN ELASTIC STRUCTURE (STA 1)

In this case, only the displacement field is concerned, while w has to be put equal to zero, and thus the whole matrix equation is reduced to:

$$[K_{uu}]\ \underline{U} = \underline{F}$$

The user has to provide the loading $\underline{F}$ and the solver computes $\underline{U}$. $\underline{F}$ has to be defined strictly for the finite element model, which means that, for example, it has to be divided by 2*pi if the model is axisymmetrical, by 2 if it is applied in a symmetry plane limiting the mesh... A **frontal assembling** is performed, coupled with a static **Gauss condensation**, and then a classical **backsubstitution** algorithm restores the whole response. Several **different loading cases** can be solved simultaneously. The user has to take care of the presence of rigid body modes, which are associated with **singularities** of the system of equations. Finally, if **distibuted loads** are applied, the user has to compute the corresponding lumped forces, using the element interpolation functions.

2.3 STATIC ANALYSIS OF A PIEZOELECTRIC STRUCTURE (STA 2)

In this case, the matrix equation is reduced to:

$$\begin{vmatrix} [K_{uu}] & \underline{K}_{uv} \\ \underline{K}_{uv}^T & K_{vv} \end{vmatrix} \begin{vmatrix} \underline{U} \\ V \end{vmatrix} = \begin{vmatrix} \underline{F} \\ -Q \end{vmatrix}$$

If the external electrical connection is a short circuit, V is equal to zero. The preceeding equation is reduced again to:

$$[K_{uu}]\ \underline{U} = \underline{F}$$

The user has to provide the loading $\underline{F}$ and the solver computes $\underline{U}$, following the same procedure as in the preceeding section. If the external electrical connection is open, Q is equal to zero and:

$$\begin{vmatrix} [K_{uu}] & \underline{K}_{uv} \\ \underline{K}_{uv}^T & K_{vv} \end{vmatrix} \begin{vmatrix} \underline{U} \\ V \end{vmatrix} = \begin{vmatrix} \underline{F} \\ 0 \end{vmatrix}$$

The user has to provide $\underline{F}$ and the solver computes $\underline{U}$ and the induced electrical potential V. Here again, attention has to be paid to rigid body modes as well as, in the second case, to the uniqueness of the electical potential. As for STA 1, several loading cases can be processed at the same time.

2.4 MODAL ANALYSIS OF AN ELASTIC STRUCTURE (MOD 1)

In this case, only the displacement field is of concern and the loading force is put equal to zero. The matrix equation is reduced to:

$$([K_{uu}] - w^2[M]) \underline{U} = \underline{0}$$

Eigenvalues and eigenvectors from this linear system of equations are the **eigenfrequencies** and **eigenmodes** of the structure.

The **ATILA** code uses frontal assembling coupled to a Guyan condensation. Thus, the user has to provide the number of **master degrees of freedom** which have to be retained in the final diagonalisation phase. To do this, he has to keep in mind that the CPU time which is reqested for the diagonalisation phase is roughly proportional to the cube of the master degree number, absolute value of this time depending upon the algorithm, the required accuracy, the computer and thc compiler option. He has also to keep in mind that accurate eigenfrequencies are obtained only for:

$$w^2 << \mathrm{Min}(K_{ii}/M_{ii})$$

where K_{ii} and M_{ii} are the diagonal terms of the stiffness and mass matrices associated with the condensed slave degrees of freedom. Thus, he has to perform a compromise between these opposing requirements. For a given master degree number, the choice of the specific degrees which have to be retained can be provided by the user or automatically selected by the code, using the Henschell's criteria.

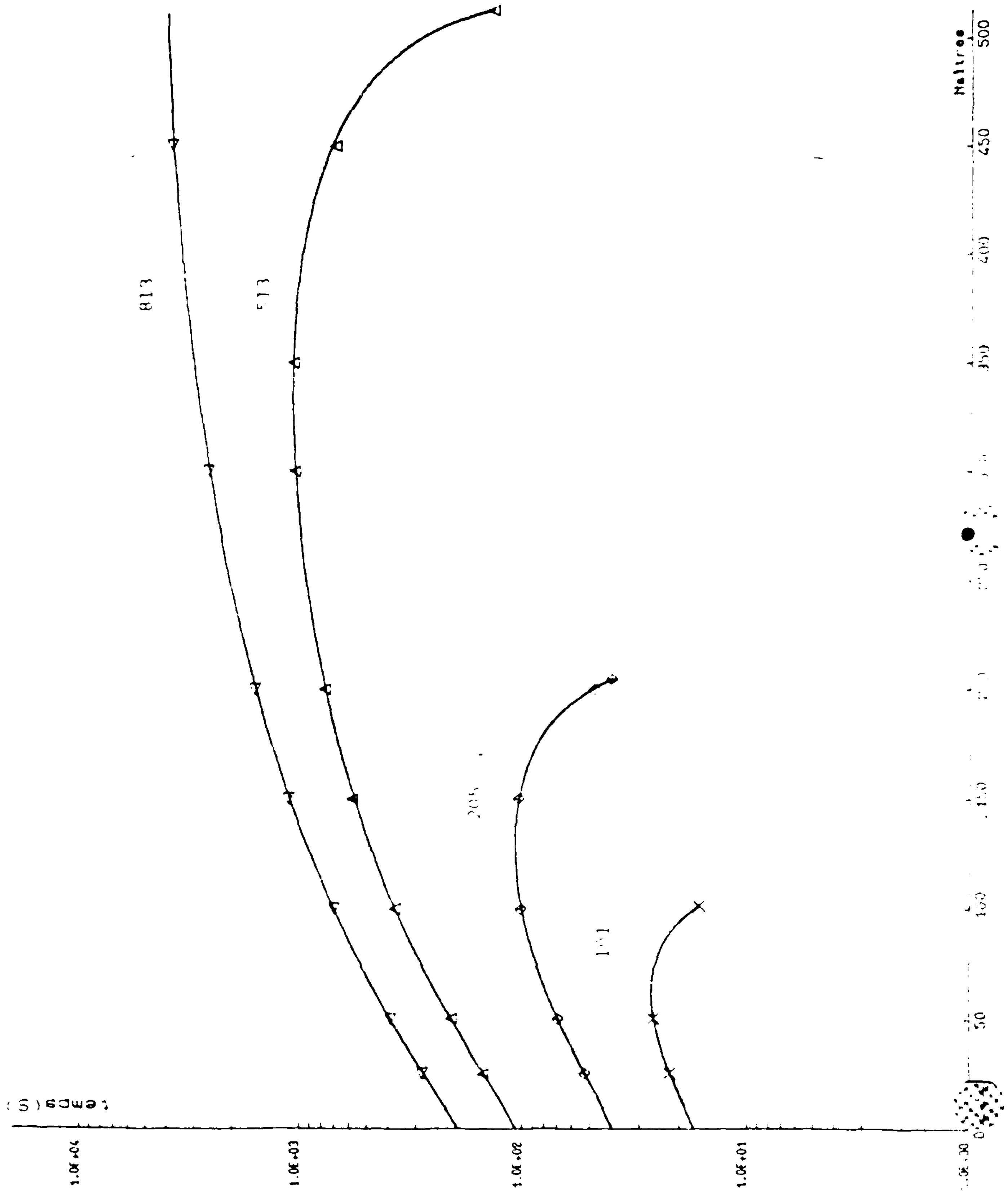
The **diagonalisation algorithm** which is used by **ATILA** relies upon the classical Householder tridiagonalisation method, coupled to a Sturm sequence. It can be processed in simple or double precision. CPU times, as obtained on an IBM 4341 Mod. 2 (4 MO), are reported on the graphic displays of pages 8 and 9. Recently, a new diagonalisation algorithm has been implemented. It provides the eigenfrequencies in double precision and the eigenvectors in single precision. Though much more efficient, it is still under test and can be only used by advanced users, following a short modification of the **ATILA** main program*. In all the cases, **ATILA** provides **all the eigenfrequencies**, the number of which is equal to the number of master degrees, and only the **first twenty eigenmodes**, if available. It provides also the **rigid body modes**, which belong to the previously mentioned first twenty modes. Upon the user's request, a **backsubstitution** routine can provide the slave degrees of freedom for any of the obtained eigenmodes.

In some cases, care has to be taken because of mixing between modes that have frequencies which are too close, as well as of matrix ill-conditioning. Successive diagonalisation with different precision or different master degree numbers can illuminate these two points.

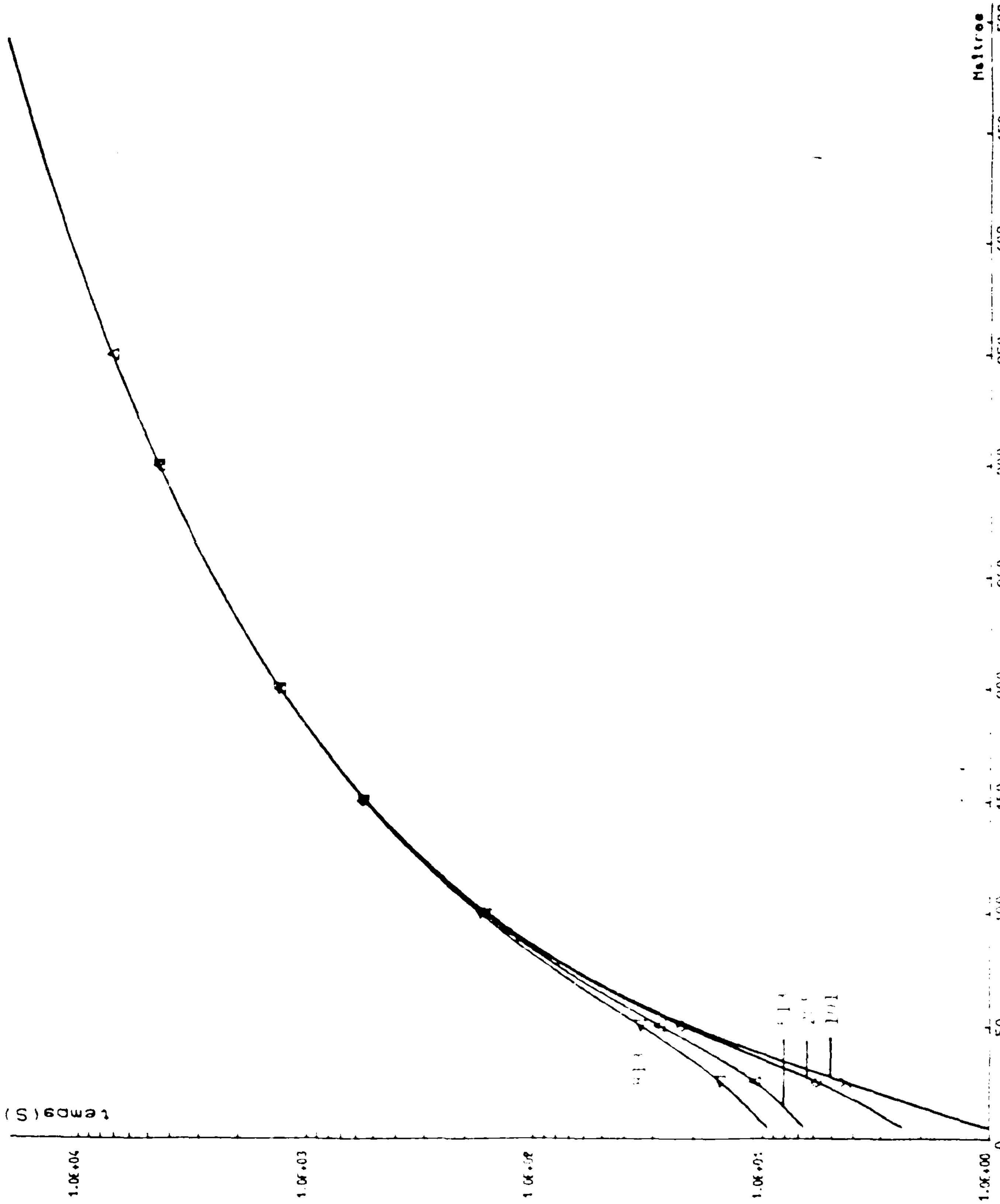
---

*. To use the new diagonalisation algorithm, the user has to edit the main program FORTRAN source, as obtained for a double precision eigenvalue computation, *replace CALL D52200 (DLIST,S, S,CPDDC,PLO,IP,DM,Z)* by CALL S52200 (DLIST,S,CPDDC,PLO,IP,Z) and delete the DM array from the DIMENSION set of instructions.

CPU TIME FOR THE ASSEMBLING PROCESS WITH RESPECT TO THE NUMBER OF MASTER DEGREES OF FREEDOM. The parameter is the total number of degrees of freedom.



CPU TIME FOR THE DIAGONALISATION PROCESS WITH RESPECT TO THE NUMBER OF MASTER DEGREES OF FREEDOM. The parameter is the total number of degrees of freedom.



2.5 MODAL ANALYSIS OF A PIEZOELECTRIC STRUCTURE (MOD 2)

In this case, the matrix equation is reduced to:

$$\begin{vmatrix} [K_{uu}] - w^2[M] & \underline{K}_{uv} \\ \underline{K}_{uv}^T & K_{vv} \end{vmatrix} \begin{vmatrix} \underline{U} \\ V \end{vmatrix} = \begin{vmatrix} \underline{0} \\ -I/jw \end{vmatrix}$$

If a resonance condition occurs, V is equal to zero and:

$$\{[K_{uu}] - w^2[M]\}\ \underline{U} = \underline{0}$$

If an antiresonance condition occurs, I is equal to zero and:

$$\{[K_{uu}] - w^2[M] - \underline{K}_{uv} \cdot \underline{K}_{uv}^T / K_{vv}\}\ \underline{U} = \underline{0}$$

In the two cases, all the information given in the preceeding section with respect to the modal analysis of an elastic structure is applicable. The only point which has to be added is that, due to the automatic static condensation of the electrical potential degrees of freedom, all these degrees of freedom are slaves.

2.6 MODAL ANALYSIS OF A CLOSED FLUID DOMAIN (MOD 3)

In this case, only the pressure field is concerned. Moreover, as zero pressure and/or zero flux boundary conditions have to be applied, the $\underline{F}'$ components which are able to appear in the equation are all zero. Thus:

$$\{[H] - w^2[M_1]\}\ \underline{P} = \underline{0}$$

All the information provided in section 2.5 with respect to the modal analysis of an elastic structure is applicable. Moreover, the user has to remember that the zero flux condition is the natural boundary condition of this finite element formulation and automatically applies to any boundary, if not superseded by another one. This modal analysis can be used to determine the irregular frequencies associated with an exterior Helmholtz integral equation.

2.7 MODAL ANALYSIS OF AN HYDROELASTIC SYSTEM (MOD 4)

In this case, structure and fluid domains exist together and interact along their interface. However, the fluid domain has to be closed, no radiation damping being included in the model. Thus, fluid surfaces which are not in contact with the solid structure have to be zero pressure or zero flux surfaces, as is the case, for example, for fluid in an open tank. Restricting the general matrix equation of section 2.1 to the displacement and pressure field, the equation which is obtained is:

$$\begin{vmatrix} [K_{uu}] - w^2[M] & - [L] \\ -a^2c^2w^2[L]^T & [H] - w^2[M_1] \end{vmatrix} \begin{vmatrix} \underline{U} \\ \underline{P} \end{vmatrix} = \begin{vmatrix} \underline{0} \\ \underline{0} \end{vmatrix}$$

Apart for a slight difference regarding the algorithm, all the information given in section 2.4 with respect to the modal analysis of an elastic structure is applicable. With respect to the **algorithm**, the above matrix has to be made symmetrical before the **ATILA** classical diagonalisation is performed. This **symmetrisation** is processed using a standard method due to Irons. However, it has to be pointed out that the symmetrisation method requires the inversion of [H] and then implies that no **fluid isopressure mode**, equivalent to structure rigid body modes, exists. Thus, at least a small part of the fluid surface has to be a zero pressure surface. Recent developments have led to the design of a new solver which avoids this difficulty, but it is still under test and can only be run by advanced users*. Moreover, during the assembling phase, as well as for an elastic structure, **Guyan condensation** is performed for both solid and fluid degrees of freedom. Care has to be given to the interface degrees of freedom which have to be kept in sufficient number. Finally, **piezoelectric parts** can be included in the structure. Then, the user has also to provide the corresponding electrical boundary conditions.

---

*. To use this new algorithm, the user has to edit the main program FORTRAN source, as obtained for this analysis, delete CALL R52300 and replace CALL R52200 by CALL S52300 with the same parameters. The new algorithm detects automatically the isopressure or rigid body modes and deals with them using a deflation method. In opposition to the standard algorithm, it can also compute the eigenmodes for mere fluid or solid domains, without interaction.

2.8 MODAL ANALYSIS OF AN ELASTIC STRUCTURE FLOODED IN AN INCOMPRESSIBLE FLUID (MOD 5)

Incompressible fluid condition is an approximation that is useful to compute the **fluid mass loading** effect on the low frequency modes of an elastic structure. Obviously, it discards any radiation effect. If it is applied to the matrix equation of the previous section, this equation becomes:

$$([K_{uu}] - w^2([M] + a^2c^2[L][H]^{-1}[L]^T)) \underline{U} = \underline{0}$$

where $a^2c^2[L][H]^{-1}[L]^T$ is the **added mass**. Then, the classical **ATILA** diagonalisation provides the structure eigenfrequencies and eigenmodes. To obtain this particular condition using **ATILA** requires that **all the fluid degrees of freedom are condensed**, this approximation producing exactly the same result as the incompressibility hypothesis*. If the fluid domain extension is **infinite**, it has to be simply **truncated** before the mesh generation. Successive truncations at various distances from the wet structure are the best convergence test.

*. After the Guyan condensation of all the fluid degrees of freedom, the matrix which has to be diagonalised is already symmetrical but the automatic program generator will provide the call for the symmetrisation subroutine because the data file corresponds to a fluid-structure modal analysis. Thus, before running **ATILA** the user must edit the FORTRAN file and delete the line CALL R52300.

2.9 HARMONIC ANALYSIS OF AN IN-AIR PIEZOELECTRIC STRUCTURE (HAR 1)

In this case, an electrical potential of given frequency is applied to an in-air free piezoelectric structure. Then, the general matrix equation of section 2.1 can be reduced to:

$$\begin{vmatrix} [K_{uu}] - w^2[M] & \underline{K}_{uv} \\ \underline{K}_{uv}^T & K_{vv} \end{vmatrix} \begin{vmatrix} \underline{U} \\ V \end{vmatrix} = \begin{vmatrix} \underline{0} \\ -I/jw \end{vmatrix}$$

To obtain this whole equation using the **ATILA** code implies that the applied electrical potential degree of freedom is not automatically condensed, which requires a **specific identification** using the boundary condition data. Then, the solver computes the displacement field $\underline{U}$, using the first line of the above matrix equation:

$$\{[K_{uu}] - w^2[M]\} \, \underline{U} = -\underline{K}_{uv} \cdot V$$

as well as the electrical impedance Z, using the second line:

$$1/Z = -jw \, (K_{vv} + \underline{K}_{uv}^T \cdot \underline{U}/V)$$

The solving algorithm is a simple Gauss elimination. In the same run, solving can be performed for **20 different excitation frequencies**. Moreover, job **restart** is possible for new frequency values without performing again the whole assembling. The **impedance value** which is provided corresponds strictly to the finite element model. Thus, the user has to divided the result by 2*pi if the model is axisymmetrical, 2 if the mesh is restricted to half of the structure by a symmetry plane, etc.

2.10 HARMONIC ANALYSIS OF AN IN-AIR PIEZOELECTRIC STRUCTURE USING THE MIXED METHOD (HAR 2)

In this case, the driving part of the structure is modelled using the plane wave approximation and a **transfer matrix method**, while the remaining inactive part of this structure is modelled using **finite element**. These two parts are in contact along a **plane interface**, which exhibits also the plane wave type displacement field. Thus, all the degrees of freedom which belong to the finite element side of the interface and correspond to a mechanical displacement normal to it have to be identical and are denoted hereafter as U_o. Then, the transfer matrix method provides, for each given frequency, a mechanical force F_o which is generated by the piezoelectric effect and is applied to the finite element part at the interface. This force can be written:

$$F_o = Z_o . U_o + F_v$$

where Z_o is a mechanical impedance and F_v contains the electrical excitation effect. Finally, the general matrix equation becomes:

$$\begin{vmatrix} [K_{uu}] - w^2[M] & \underline{K}_o \\ \underline{K}_o^T & K_{oo} \end{vmatrix} \begin{vmatrix} \underline{U} \\ U_o \end{vmatrix} = \begin{vmatrix} \underline{0} \\ F_o \end{vmatrix}$$

where $[K_{uu}]$ and $[M]$ correspond to all the degrees of freedom apart from U_o. The user has to select the excitation frequencies and to specify, using the correct boundary condition data, the degree of freedom U_o. The solving algorithm is a simple Gauss elimination, and the code provides the displacement field in the finite element part of the structure. **Twenty different excitation frequencies** can be provided for the same run. Moreover, as in section 2.9, job **restart** is possible with new frequency values. This mixed method is generally well suited to the modelling of Tonpilz type transducers and allows large savings of CPU time .

2.11 RADIATED PRESSURE COMPUTATION FOR A GIVEN DISPLACEMENT FIELD (HAR 3)

In this case, a vibrating structure radiates an acoustic wave in an infinite medium and the vibration displacement is assumed to be known for each given frequency. The general matrix equation from section 2.1 can be reduced to:

$$\{[H] - w^2[M_1]\}\ \underline{P} = ac^2\underline{F}' + a^2c^2w^2[L]^T.\underline{U}$$

Then, if monopolar or dipolar damping is used, following the third equation of section 2.1, the damping term can be written under the form:

$$ac^2\underline{F}' = [G]\ \underline{P}$$

where [G] stands for a linear complex frequency dependent operator. Thus:

$$\{[H] - w^2[M_1] - [G]\}\ \underline{P} = a^2c^2w^2[L]^T.\underline{U}$$

The user has to define $\underline{U}$ and the given excitation frequencies. **To define U**, he has to write a small subroutine, named **FABU***, which contains the displacement value for each interface solid degree of freedom, and has to be added to the main program. Only **one displacement field** can be defined at a time and this field has to be real (phase reference). **Ten successive frequencies** can be processed in the same run and, as for the previous section, **restart** is available. The solver splits the previous complex equation into two real equations. Then, a matrix inversion followed by a Gauss elimination is performed. For each frequency, **ATILA** provides the **real and imaginary parts of the pressure field**, printed in two successive tables.

To obtain accurate results, the user has to give a great deal of attention to the damping element choice. If **monopolar** damping is used, the external boundary surface has to be centered at the acoustic center and it has to be in the farfield. If **dipolar** damping is used, the external boundary surface has also to be centered at the acoustic center but, generally, it can be deeply in the nearfield. However, in this second case, farfield information is only available after an appropriate postprocessing. Dipolar elements can only be used for axisymmetrical models. Three-dimensional dipolar element design is in progress.

Degrees of freedom which belong to the exterior boundary surface are automatically master degrees. Those which belong to the interface can be slave degrees but the user has to keep a sufficient number of them as master degrees. Finally, care has to be taken to keep the maximal distance between two successive fluid master nodes in the propagation direction less than one quarter of a wavelength.

---

*. The FORTRAN subroutine which has to be added at the end of the main program, to provide the given displacement field $\underline{U}$, is FABU (CPDDC,U,IOLE,IP). It is called by the solving subroutine R52400 during the run. IOLE and IP are control integers which are defined in R52400, as well as the CPDDC array. In return, FABU provides all the components of $\underline{U}$, numbered in the ascending order. $\underline{U}$ has to contain IDT components, where IDT is the total number of degrees of freedom, but, in fact, only the mechanical degrees of freedom are concerned. The following lines are provided as a standard example:

```
SUBROUTINE FABU(CPDDC,U,IOLE,IP)
DIMENSION CPDDC(IP,*), U(*)
U(1) = ...
U(2) = ...
..........
..........
RETURN
END
```

2.12 HARMONIC ANALYSIS OF A PIEZOELECTRIC RADIATING STRUCTURE (HAR 4)

This is the general case. Using the operator [G] defined in the previous section 2.11, the matrix equation is:

$$\begin{vmatrix} [K_{uu}] - w^2[M] & \underline{K}_{uv} & -[L] \\ \underline{K}_{uv}^T & K_{vv} & \underline{0}^T \\ -a^2c^2w^2[L]^T & \underline{0} & [H]-w^2[M_1]\end{vmatrix} \begin{vmatrix}\underline{U}\\ V\\ \underline{P}\end{vmatrix} = \begin{vmatrix}\underline{0}\\ -I/jw \\ [G]\underline{P}\end{vmatrix}$$

Then, it can be transform into:

$$\begin{vmatrix} [K_{uu}] - w^2[M] & -[L] \\ -a^2c^2w^2[L]^T & \{[H]-w^2[M_1]-[G]\}\end{vmatrix} \begin{vmatrix}\underline{U}\\ \underline{P}\end{vmatrix} = \begin{vmatrix}-\underline{K}_{uv}.V\\ \underline{0}\end{vmatrix}$$

and:

$$1/Z = -jw\,(\underline{K}_{uv}^T.\underline{U} + K_{vv}.V)$$

The user provides the excitation frequencies and, using specific boundary condition data, defines the degree of freedom that is associated with the applied electrical potential. The solver computes, using matrix inversion and Gauss condensation, the **real and imaginary parts of the displacement and pressure fields**. Then, it computes the electrical impedance. Real and imaginary parts are provided in two successive tables. **Ten successive excitation frequencies** can be processed in the same run, and, as in the previous section, **restart** is available.

Information given in the previous section and regarding damping elements, nearfield or farfield conditions, boundary surface and interface degrees of freedom remain available for this section and are not duplicated. Information given in section 2.9 regarding the electrical impedance is also valid.

2.13 HARMONIC ANALYSIS OF A PIEZOELECTRIC RADIATING STRUCTURE USING THE MIXED METHOD (HAR 5)

This case is a mixing of the previous cases reported in sections 2.10 and 2.12. Using the corresponding notations, the matrix equation becomes:

$$\begin{vmatrix} [K_{uu}] - w^2[M] & \underline{K}_o & -[L] \\ \underline{K}_o^T & K_{oo} & \underline{0}^T \\ -a^2c^2w^2[L]^T & \underline{0} & [H] - w^2[M_1] \end{vmatrix} \begin{vmatrix} \underline{U} \\ U_o \\ \underline{P} \end{vmatrix} = \begin{vmatrix} \underline{0} \\ F_o \\ [G]\underline{P} \end{vmatrix}$$

The user provides the excitation frequencies and, using specific boundary condition data, defines the degree of freedom that is associated with the normal displacement to the interface. The solver computes, using matrix inversion and Gauss condensation, the **real and imaginary parts of the displacement and pressure fields**. Real and imaginary parts are provided in two successive tables. **Ten successive excitation frequencies** can be processed in the same run, and, as in the previous section, **restart** is available.

Information given in section 2.11 regarding damping elements, nearfield or farfield conditions, boundary surface and interface degrees of freedom remain available for this section and are not duplicated.

```
 AAAAAAAA    TTTTTTTTTT   II   LL          AAAAAAAA
AA      AA       TT       II   LL         AA      AA
AA      AA       TT       II   LL         AA      AA
AAAAAAAAAA       TT       II   LL         AAAAAAAAAA
AA      AA       TT       II   LL         AA      AA
AA      AA       TT       II   LL         AA      AA
AA      AA       TT       II   LLLLLLL    AA      AA
```

FINITE ELEMENT MODELING

OF

PIEZOELECTRIC TRANSDUCERS

AUGUST 1987

CHAPTER 3
DESCRIPTION OF A DATA FILE

THE FINITE ELEMENT CODE **ATILA** HAS BEEN DESIGNED BY THE ACOUSTICS LABORATORY OF THE "**INSTITUT SUPERIEUR D'ELECTRONIQUE DU NORD**" (ISEN. LILLE. FRANCE) FOR THE "**GROUPE D'ETUDES ET DE RECHERCHE EN DETECTION SOUS-MARINE**" (GERDSM. DCAN. FRENCH NAVY. TOULON). THE SOFTWARE ENGINEERING IS PROVIDED BY THE **SINAPTEC** COMPANY (LILLE. FRANCE).

3.1 INTRODUCTION

This section is placed first in the data preparation part of this manual for pedagogical purposes only. In fact, in most cases, an automatic data generation can be processed using the **MOSAIQUE** code and a more simple initial data file. This automatic data generation is described in the **chapter 6**. When used first, **MOSAIQUE** provides data files which are identical to the files described hereafter.

In the general case, an **ATILA** data file includes five different sections:

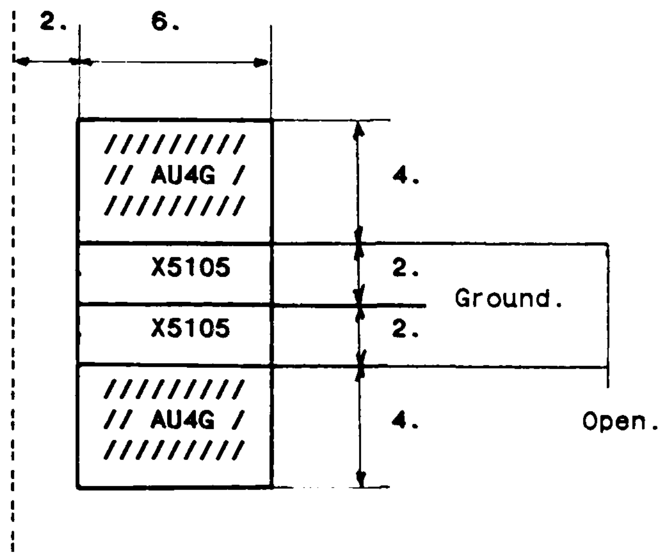
- **the header**, which contains only comments and describes the job under consideration,
- **a set of entries**, which allows, among other things, to describe the mesh and the properties of the materials,
- **a set of data**, which describes the loading and boundary conditions,
- **a set of data**, which describes the plane wave part of the structure in the case of a mixed model,
- **a set of data**, which defines the graphic displays.

All data, except for the angles, must be given using **MKS** units or any coherent unit system which is deduced from the MKS system. The angle values are always given in degrees.

In this chapter, we always refer to a **simple illustrative example** and give the corresponding version of each entry or data set. This example is the modal analysis of an elementary axisymmetrical piezoelectric structure, made up of two ceramic rings and two thick electrodes, under electrical antiresonance conditions. This structure has a symmetry plane which is normal to the axis and the modes which have to be computed are symmetrical with respect to this plane. Five full eigenvectors are requested. The structure, its material properties and the corresponding mesh are described on the following two pages. Other examples are given at the end of the manual.

Test example description.

Geometry.



SYMMETRY AXIS

Dimensions are in centimeters.

Material properties.

AU4G (aluminium alloy).

.Young's modulus: 0.714E+11 Pa
.Poisson's ratio: 0.344
.Density: 2780 kg/m^3

X5105 (piezoelectric ceramic material).

.Polarization: axial (up for the top ceramic ring and down for the bottom ceramic ring).
.Elastic constants (MKS units, constant electric field):

S11: 0.114E-10 S33: 0.126E-10
S12: -0.339E-11 S13: -0.410E-11
S44: 0.510E-10

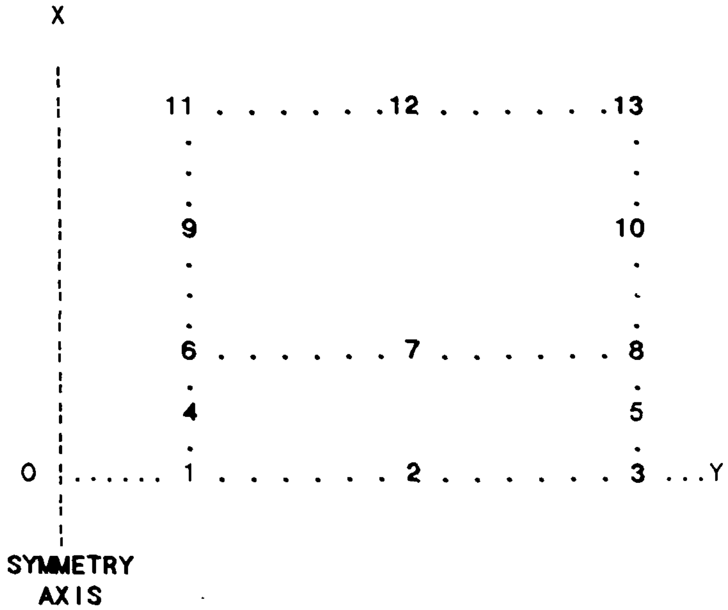
.Piezoelectric constants (MKS units):

d33: 0.208E-09 d31: -0.955E-10
d15: 0.491E-09

.Dielectric constants (MKS units, constant strain):

EP11: 0.667E-08 EP33: 0.687E-08

Mesh.



Due to simple symmetry arguments, the mesh is limited to one half of the structure. Thus, the OY axis is the intersection of the symmetry plane with the plane of the figure.

3.2 THE HEADER

The header includes comments which describe the job under consideration. The whole header is automatically copied on the first page of every listing which is related to the job. The first line of the header, generally a title, is automatically repeated ahead of each listing page. To define the header, each line begins with the symbol *. The number of lines in a header is unlimited.

EXAMPLE.

```
+ + + + + + + + + + + + + + + + + + + + + + + + +
+ *... MODAL ANALYSIS . PIEZOELECTRIC STRUCTURE .
+ *..............................................
+ *... ANTIRESONANCE CASE .......................
+ *... AXISYMMETRICAL MODEL .....................
+ *... 1/2 OF THE STRUCTURE (SYMMETRY PLANE) ....
+ *..............................................
+ *... NUMBER OF ELEMENTS: 2 ....................
+ *... NUMBER OF NODES: 13 ......................
+ *... NUMBER OF DOF: 26 ........................
+ *... NUMBER OF MASTER DOF: 23 .................
+ *... DOUBLE PRECISION .........................
+ *..............................................
+ *... DATA FILE WRITTEN BY: ....................
+ *... DATE .....................................
+
+
```

3.3 THE ENTRIES

The various entries which are described hereafter are listed in a given order, which corresponds to a classical use but is not mandatory. These entries have to be written using a simple **free format**, described below, which allows an easy reading of the file. An entry can be associated to **given peculiar words**, named **attributes**, which have to be written on the entry line, or to **given numerical values**, named **parameters**, which have to be written on the following line. Entries being read in free format, all the attributes and parameters have to be provided, even if they are dummy or if their numerical value is zero.

3.3.1 DESCRIPTION OF THE FREE FORMAT.

Control characters.

. [ ] and [,] separate two words on the same line. They have to be used to separate an entry from its following attribute, to separate two following attributes or to separate two following parameters.
. [/] and [=] separate two following lines (carriage return). As well as the carriage return, they have to be used to separate an entry from its following parameter.
. [?] deletes the preceeding characters of the line,
. [*] when typed at the beginning of a new line, means that this line is a comment. This line is not printed on a listing. Such a comment has not to be confused with the header.
. [*............*] allows the writing of a comment in an entry line. This comment is not printed on a listing.

Real number syntax.

The following syntaxes, given as examples, are available:

```
.   1.22E+02            .   -1.22E2
.   -1E-3               .    1.22
.   -1.                 .   -1
```

Examples.

The following sequences are equivalent to define the given example: one alphanumeric datum (alpha) on the first line, two integers (1,2) on the second line, three real numbers (1.,2.,3.) on the third line:

```
. alpha          . alpha=1,2        . alpha/1 2=1.,2. 3.
  1,2              1. 2.,3.
  1.,2.,3.
```

3.3.2 THE ENTRY DIRECTORY.

NAME	COMP.	ATTRIBUTES OR PARAMETERS	P
ANALYSIS	YES	STATIC, MODAL or HARMONIC	7
CLASS	NO	AXISYMMETRICAL, PSTRESS or PSTRAIN	7
NLOAD	YES	NLO	7
PRECISION	NO	SINGLE or DOUBLE	8
LCPDDC	YES	NLC	8
NODES	YES	node coordinates (end of the set: dummy line)	9
NEWAXES	NO	CARTESIAN, CYLINDRICAL or SPHERICAL XO YO ZO OZZ OYY OXX	10
SCALE	NO	EX EY EZ	11
ELEMENTS	YES	Rxxxxx MATERx NGx Element topology (end of the set: dummy line)	12
MATERIALS	YES	MATERx Properties of the material. (end of the set: dummy line)	14
GEOMETRY	NO	NGx Geometrical properties. (end of the set: dummy line)	17
GEOMETRY POLARIZAT.	NO	CARTESIAN, CYLINDRICAL or SPHERICAL. NGx Euler angles, coordinates. (end of the set: dummy line)	18
REDUCTIONS	YES	I N MDF	24
FREQUENCIES	NO	Frequency values.	25
RADIATION	NO	MONOPOLAR or DIPOLAR.	25
PRINTING	NO	NPR	25
END	YES		26

COMP. means that the entry is compulsory.

3.3.3 DESCRIPTION OF THE ENTRIES.

In this section, the use of the symbol ● in front of the entry name means that this entry is **compulsory** and has to be used in every job. The other entries have to be used only in specific jobs or are optional.

● <u>ANALYSIS</u> **STATIC** or **MODAL** or **HARMONIC**

This entry defines the kind of analysis which is requested in the job (see chapter 2, theoretical background) and is **compulsory**.

Test example.

```
+
+ ANALYSIS  MODAL ...........
+
```

<u>CLASS</u> **AXISYMMETRICAL** or **PLSTRESS** or **PLSTRAIN**

This entry specifies, if necessary, that the analysis corresponds to an axisymmetrical, a plane stress or a plane strain model. It is only needed if 2D elastic elements are used, but then is compulsory.

Test example.

```
+
+ CLASS  AXISYMMETRICAL .....
+
```

● <u>NLOAD</u>
NLO

This entry is **compulsory** and provides NLO which is:

. the number of **loading cases** for a static analysis,
. the number of full **eigenvectors** which are required in the case of a modal analysis,
. the number of **given frequency values** in the case of an in-air harmonic analysis,
. twice the number of **given frequency values** in the case of an harmonic analysis of a radiating structure.

Test example.

```
+
+ NLOAD = 5 .................
+
```

PRECISION SINGLE or DOUBLE

This entry is only used if an eigenvalue computation is run after the assembling and specifies the precision of this computation. The **default** answer is single.

Test example.

```
+
+ PRECISION DOUBLE ..........
+
```

- **LCPDDC**
  NLC

The CPDDC array contains the node coordinates (X, Y, Z) and the numbers of the node degrees of freedom (translations UX, UY, UZ, rotations NX, NY, NZ, pressure P and electrical potential V). These data are listed, with one row per node, in the following order:

X, Y, Z, UX (or P), UY, UZ, NX (or V), NY, NZ

The **LCPDDC** entry is **compulsory** and provides **NLC**, which is the number of terms needed in one row. Thus, in the case of an axisymmetrical analysis of an elastic structure **NLC** equals 5, while in an axisymmetrical analysis of a piezoelectric structure **NLC** equals 7 and in an analysis of a plate **NLC** equals 8. If useless intermediate columns are introduced, due to the given value of NLC, the corresponding degrees of freedom will have to be deleted later. This case happens, as an example, for the degrees of freedom included in columns 3, 4 and 5 in the plate case.

Test example.

```
+
+ LCPDDC ....................
+ 7 .........................
+
```

- **NODES**

```
X1 Y1 Z1
X2 Y2 Z2
X3 Y3 Z3
.
.
.
.
XN YN ZN
.DUMMY..
```

This entry provides the node coordinates, listed with one row per node, in the node number ascending order. The list has to be ended by a dummy line and the entry is **compulsory**. If, for any reason, the node coordinate list is split in several successive sets, each set requires a new entry **NODES** and has to be finished by a new dummy line.

WARNING

> **When the model includes both structure and fluid domains, nodes have to be set in the following order:**
>
> SOLID - FLUID - RADIATING.
>
> **For axisymmetrical models, the OX axis is necessarily the symmetry axis.**

Test example.

```
+
+ NODES .....................
+ 0.00  0.02 ...............
+ 0.00  0.05 ...............
+ 0.00  0.08 ...............
+ 0.01  0.02 ...............
+ 0.01  0.08 ...............
+ 0.02  0.02 ...............
+ 0.02  0.05 ...............
+ 0.02  0.08 ...............
+ 0.04  0.02 ...............
+ 0.04  0.08 ...............
+ 0.06  0.02 ...............
+ 0.06  0.05 ...............
+ 0.06  0.08 ...............
+ ..........................
+
+
```

NEWAXES CARTESIAN or CYLINDRICAL or SPHERICAL
XO YO ZO OZZ OYY OXX

The initial coordinate system in which the node coordinates are given (**default** system) is cartesian and its origin and axes are the absolute origin and axes. The entry **NEWAXES** allows it to be modified.
If the new system is cylindrical (spherical), the node coordinates X, Y, Z, associated to the following **NODES** entry(ies) become the cylindrical (spherical) coordinates R, THETA, Z (R, THETA, PHI). Moreover, XO, YO, ZO are the coordinates of the origin of the new system, while OZZ, OYY, OXX are the angle values of the Euler rotations which transform the initial axes in the new axes. The first rotation OZZ is around the OZ axis of the initial system. The second rotation OYY is around the OY axis provided by the first rotation. The third rotation OXX is around the OX axis provided by the second rotation. Angle values have to be given in degrees.
This entry can be used **several times** during the description of the mesh, its parameters being always defined with respect to the absolute system. To **come back** to the absolute system, it has to be used with the CARTESIAN attribute, the parameters being equal to zero.

Test example.

```
+
+ NODES .....................
+ 0.00  0.02 ................
+ 0.00  0.05 ................
+ 0.00  0.08 ................
+ 0.01  0.02 ................
+ 0.01  0.08 ................
+ 0.02  0.02 ................
+ 0.02  0.05 ................
+ 0.02  0.08 ................
+ ...........................
+ NEWAXES  CARTESIAN ........
+ 0.02  0.0 0.0 0.0 0.0 0.0 .
+ NODES .....................
+ 0.02  0.02 ................
+ 0.02  0.08 ................
+ 0.04  0.02 ...............
+ 0.04  0.05 ...............
+ 0.04  0.08 ...............
+ ..........................
+
+
```

Although this data file is correct, it is quite evident that it has no specific usefulness with respect to this example.

SCALE
EX EY EZ

If necessary, this entry defines scale factors (EX, EY, EZ) which are applied to the node coordinates included in the following **NODES** entry(ies), during the description of the mesh. To switch off this scaling, the **SCALE** entry has to be used with the parameters 0, 0, 0 or 1, 1, 1.

Test example.

```
+
+ NODES .....................
+ 0.00  0.02 ................
+ 0.00  0.05 ................
+ 0.00  0.08 ................
+ 0.01  0.02 ................
+ 0.01  0.08 ................
+ 0.02  0.02 ................
+ 0.02  0.05 ................
+ 0.02  0.08 ................
+ ...........................
+ NEWAXES  CARTESIAN ........
+ 0.02  0.0 0.0 0.0 0.0 0.0 .
+ SCALE .....................
+ 2.0 1.0 1.0 ...............
+ NODES .....................
+ 0.01  0.02 ................
+ 0.01  0.08 ................
+ 0.02  0.02 ................
+ 0.02  0.05 ................
+ 0.02  0.08 ................
+ ...........................
+
+
```

This example is a simple combination of the entries **NEWAXES** and **SCALE**. It allows the modification of the thickness of the electrode by simply modifying the scale factor (2.00 in the above file).

- **ELEMENTS**

```
RAAAAA MATERA NGA
N11 N12 N13 .   .   .   . N1P
N21 N22 N23 .   .   .   . N2P
 .   .   .  .   .   .   . .
NK1 NK2 NK3 .   .   .   . NKP
........... DUMMY ...........
RBBBBB MATERB NGB
N11 N12 N13 .   .   .   .   .   .   .   .   . N1Q
N21 N22 N23 .   .   .   .   .   .   .   .   . N2Q
 .   .   .  .   .   .   .   .   .   .   .   . .
NL1 NL2 NL3 .   .   .   .   .   .   .   .   . NLQ
........... DUMMY ...........
.
.
.
.
........... DUMMY ...........
RZZZZZ MATERZ NGZ
N11 N12 N13 .   .   N1R
N21 N22 N23 .   .   N2R
 .   .   .  .   .   .
NM1 NM2 NM3 .   .   NMR
........... DUMMY ...........
........... DUMMY ...........
```

This entry provides information about the elements. It is **compulsory**. Its **parameters are defined set by set**, each set corresponding to successive elements of the same type. The various types of elements, their topologies and their corresponding material and geometrical properties are described in **chapter 4**.

For each set of parameters, the first line contains the name of the element type, corresponding to RAAAAA, RBBBBB, ..., RZZZZZ in the above entry example, the name of the corresponding material data set, denoted MATERx (see the entry **MATERIALS** for more details), and the number of the corresponding geometrical property set, denoted NGx (see the entry **GEOMETRY** for more details). Then, each following line contains the actual numbers of the element nodes, given in a compulsory order which corresponds to the element topology (one or two lines per element, depending upon the number of nodes belonging to it, the first line being terminated by the character & if a second line is needed). Finally, the set has to be closed by a dummy line. Moreover, the whole entry parameter list has also to be closed by a dummy line. Thus, the two last lines of this entry are always two *dummy* lines.

From the computational point of view, elements are assembled in the sequential order of this entry parameter list. Thus, in most of the cases, a great deal of attention has to be given to this order, which has an important effect on the array sizes and the CPU time requested by the job.

Test example.

```
+
+ ELEMENTS .................
+ R37521 X51 1 .............
+ 1 3 6 8 2 4 5 7 ..........
+ .........................
+ R36210 AU4G 2 ............
+ 6 8 11 13 7 9 10 12 ......
+ .........................
+ .........................
+
+
```

The above numberings of the nodes can be easily obtained using the corresponding element descriptions in **chapter 4**. "X51" and "AU4G" are parameters which refer to the **MATERIALS** entry. "1" and "2" are parameters which refer to the **GEOMETRY** entry. Their meaning is discussed in the two following sections.

- **MATERIALS**

```
MATER1
X11 X12 X13
MATER2
X21 X22 X23 .   . X2P
.
.
.
MATERN
XN11 XN12 XN13 XN14 XN15 XN16 &
.    .    .    .    .    .    &
.    .    .    .    .    .    &
XNM1 XNM2 XNM3 XNM4 XNM5 XNM6 &
......... DUMMY LINE ..........
```

This entry provides information about the material properties. It is **compulsory** and has to be closed by a dummy line. Its parameters are defined by set. Each parameter set corresponds to a given material and is identified by a name, selected by the user and containing less than 8 characters (MATER1, MATER2, ...,MATERN in the above example). This name links a set of elements from the previous **ELEMENTS** entry to the given material. The list of the requested properties is provided in the element description, in **chapter 4**. Thus, as examples, this list contains Young's modulus, Poisson's ratio and density in the case of a three-dimensional elastic element, and it contains the whole elastic, piezoelectric and dielectric tensors in the case of a piezoelectric element. If a line is insufficient to describe the properties of a given material, it can be extended using the character &. The order in which the physical properties have to be written is compulsory and is given below.

. **elastic material:**

E NU RO

where E is Young's modulus, NU is Poisson's ratio and RO is the density.

. **piezoelectric material:**

```
0.0  0.0  RO   0.0  0.0  0.0
S11  S12  S13  S14  S15  S16
S21  S22  S23  S24  S25  S26
S31  S32  S33  S34  S35  S36
S41  S42  S43  S44  S45  S46
S51  S52  S53  S54  S55  S56
S61  S62  S63  S64  S65  S66
d11  d12  d13  d14  d15  d16
d21  d22  d23  d24  d25  d26
d31  d32  d33  d34  d35  d36
EP11 EP12 EP13 0.0  0.0  0.0
EP21 EP22 EP23 0.0  0.0  0.0
EP31 EP32 EP33 0.0  0.0  0.0
```

where [S] is the constant electric field elastic tensor, [d] is the piezoelectric tensor and [EP] is the constant strain dielectric tensor.

. **fluid material:**

COMP .0 RO
where COMP is the modulus of compression and RO is the density.

The material property values of classical or more frequently used materials can be written in a special file, named **MATER.STD**. If it is the case, these materials need not be described in the **MATERIALS** entry and their names can be used directly as parameters in the **ELEMENTS** entry. If one of the names belonging to the **MATER.STD** file is used in the **MATERIALS** entry, the new property values which are listed as parameters will automatically replace the old values in the file and will be used by the code during the run. In this case, a warning will be printed in the listing file. Finally, if a new material name is defined in the **MATERIALS** entry, this name and the corresponding property values will be automatically added to the **MATER.STD** file.
The **MATER.STD** file can be **edited** by the user for any modification, such as adding a new material or correcting old property values. Its first line contains a control integer (generally 80, which is the *number of columns*), the date of the last edition and its **total number of lines**. The following lines contain the material property values, listed as defined above (1 line for elastic or fluid materials, 13 lines for piezoelectric materials). Each of these following lines starts with two special fields, containing respectively the material name and the number of lines set apart for this material (as examples: STEEL 1 or CERAMIC 13 ...). If this file is edited, the user has to take care of the final total number of lines and, if needed, to correct its value on the first line of the file. An opening error can take place during the following run of **ATILA** if this number is wrong.

Test example.

```
+
+ MATERIALS ........................................
+ AU4G .............................................
+ .714E+11 .344 2780. ..............................
+ X51 ..............................................
+ 0.0 0.0 7350. 0.0 0.0 0.0 .......................
+  0.114E-10 -0.339E-11 -0.410E-11 0.0 0.0 0.0 & .
+ -0.339E-11  0.114E-10 -0.410E-11 0.0 0.0 0.0 & .
+ -0.410E-11 -0.410E-11  0.126E-10 0.0 0.0 0.0 & .
+  0.0 0.0 0.0 0.510E-10 0.0 0.0 & ..............
+  0.0 0.0 0.0 0.0 0.510E-10 0.0 & ..............
+  0.0 0.0 0.0 0.0 0.0 0.296E-10 & ..............
+  0.0 0.0 0.0 0.0 0.491E-09 0.0 & ..............
+  0.0 0.0 0.0 0.491E-09 0.0 0.0 & ..............
+ -0.955E-10 -0.955E-10  0.208E-09 0.0 0.0 0.0 & .
+  0.667E-08 0.0 0.0 0.0 0.0 0.0 & ..............
+  0.0 0.667E-08 0.0 0.0 0.0 0.0 & ..............
+  0.0 0.0 0.687E-08 0.0 0.0 0.0 ................
+ ..................................................
+
```

GEOMETRY
NG1
X11 X12
NG2
X21 X22 .   . X2P
.
.

.
NGM
XM1 XM2 .   .   . XMQ
.... DUMMY LINE .....

This entry provides information about the geometrical properties. It must be closed by a dummy line. Its parameters are defined set by set, each parameter set corresponding to a given geometry and being identified by a number selected by the user (NG1, NG2,...,NGM in the above example). For a given geometry, the list of the requested properties is provided in the element description, in **chapter 4**. Thus, as examples, this list contains the thickness in the case of a plate element, and it contains the radius of curvature in the case of a damping element. The order in which the geometrical properties have to be written is compulsory.

Test example.

In this case, the correct entry for geometrical properties is the **GEOMETRY POLARIZATION** entry which is described in the next section. In fact, its format is identical to the **GEOMETRY** entry format, and thus the example can be considered as common to the two entries.

GEOMETRY POLARIZATION STRING
NG1
X11 X12
NG2
X21 X22 . . X2P
.
.
.
NGM
XM1 XM2 . . . XMQ
.... DUMMY LINE .....

This entry provides information about the geometrical properties when piezoelectric elements are present in the mesh, and it is compulsory in this case. The **GEOMETRY** entry features are all relevant to this new entry and must be used for non-piezoelectric elements. However, for piezoelectric elements, specific parameters have to be added to define the polarization type (cartesian, cylindrical or spherical) and orientation.The polarization type is specified by replacing **STRING** by **CARTESIAN**, **CYLINDRICAL** or **SPHERICAL** in the entry name.

. If **STRING=CARTESIAN**, the polarization is **homogeneous** in the whole element and the natural axes OX_1, OX_2 and OX_3 have to be specified with respect to the global axes OX, OY and OZ (OX_3 is the polarization axis in the classical case of a ceramic material). To do this, the user has to provide three Euler angles, named ALPHA, BETA and GAMMA, which transform OXYZ into $OX_3X_1X_2$ and are defined as follows (see figure 1):

- ALPHA is the angle of the first rotation, around OZ, which transforms the OXYZ system into a OX'Y'Z system such that OX' belongs to the OZX_3 plane,
- BETA is the angle of the second rotation, around OY', which transforms the OX'Y'Z system into a $OX_3Y'Z'$,
- GAMMA is the angle of the third rotation, around OX_3, which transforms the $OX_3Y'Z'$ system into the $OX_1X_2X_3$ system. Its value is dummy in the particular case of a ceramic material (6mm symmetry class).

The entry parameters are ALPHA, BETA, GAMMA, listed in this compulsory order. Attention has to be given to the exact meaning of this "cartesian" polarization. In fact, if the polarization is homogeneous in a three-dimensional element, it is physically homogeneous in the corresponding part of the material. On the other hand, if the polarization is homogeneous and its direction orthogonal to the symmetry axis in the case of an axisymmetrical element, this polarization is physically cylindrical in the corresponding part of the material.

. If **STRING=CYLINDRICAL**, the polarization is **radial around a given axis O'Z'** in the whole element. The user has first to specify a O'X'Y'Z' system which contains the O'Z' axis. To do this, he provides the O' coordinates (XO, YO, ZO) as well as the Euler angles (ALPHA, BETA, GAMMA) which transform the OXYZ system into the O'X'Y'Z' system and are defined as follows (see figure 2a):

- ALPHA is the angle of the first rotation, around OZ, which transforms the OXYZ system into a OxyZ system such that Ox belongs to the OZX' plane,
- BETA is the angle of the second rotation, around Oy, which transforms the OxyZ system into a OX'yz,
- GAMMA is the angle of the third rotation, around OX', which transforms the OX'yz system into the OX'Y'Z' system.

It is self evident that the choice of the O'X' (or O'Y') axis is arbitrary and has to be the simplest. With these new reference axes, the angle of the rotation around O'Z' which brings the O'X' axis to coincidence with the natural MX_3 axis, for each point M which is concerned, is directly computed by the code (see figure 2b). Finally, the user must also provide the angle DELTA which brings, for every point M, the MZ' axis into coincidence with the MX_2 natural axis (see figure 2c). The value of DELTA is dummy in the case of a ceramic material (6mm symmetry class). The entry parameters are ALPHA, BETA, GAMMA, XO, YO, ZO, DELTA, listed in this compulsory order. Attention has to be given to the exact meaning of this "cylindrical" polarization. In fact, if the polarization is cylindrical in a three-dimensional element, it is physically cylindrical in the corresponding part of the material. On the other hand, if the polarization is cylindrical around an axis normal to the plane of an axisymmetrical element and intersecting the symmetry axis at the point O, this polarization is physically spherical around O in the corresponding part of the material.

. If **STRING=SPHERICAL**, the polarization is **radial around a given point O'** in the whole element. The user has first to specify a O'X'Y'Z' system centered on the origin O'. To do this, he provides the O' coordinates (XO, YO, ZO) and the Euler angles which transform the OXYZ system into the O'X'Y'Z' system (ALPHA, BETA, GAMMA) and are defined as for the cylindrical case (see figure 3a). It is self evident that the choice of these new axes is arbitrary and has to be the simplest. With these new reference axes, the angles of the rotations which bring the O'X' axis into coincidence with the natural MX_3 axis, for each point M which is concerned, are computed by the code (see figure 2b). Finally, the user has also to provide the angle DELTA which brings, for every point M, the MZ' axis into coincidence with the natural MX_2 axis (see figure 2c). The value of DELTA is dummy in the case of a ceramic material (6mm symmetry class). The entry parameters are ALPHA, BETA, GAMMA, XO, YO, ZO, DELTA, listed in this compulsory order.

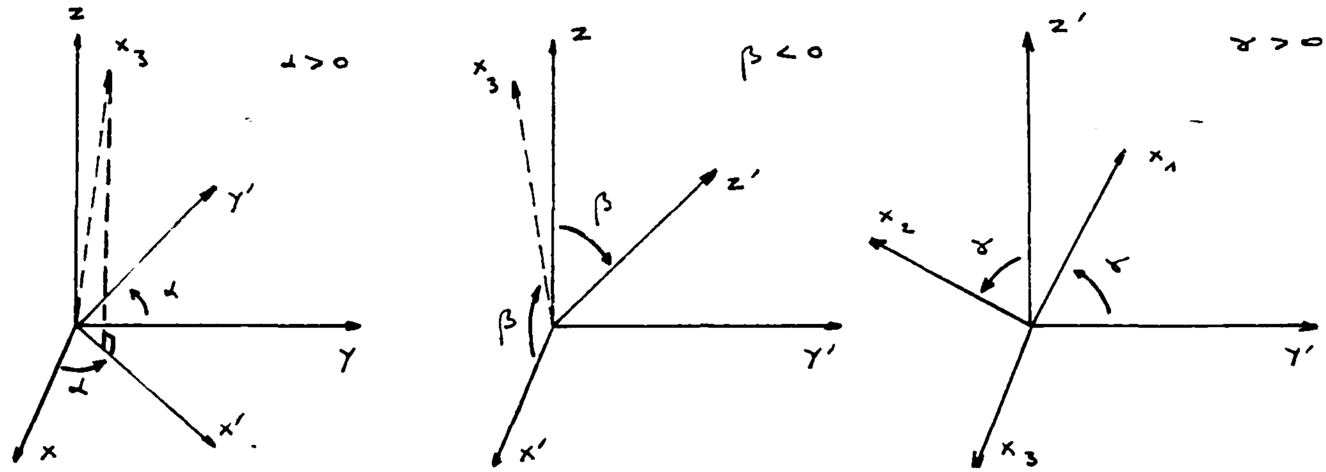
Test example.

```
+
+ GEOMETRY POLARIZATION CARTESIAN ...........
+ 1 .....................................
+ 0.0 0.0 0.0 0.0 0.0 0.0 0.0 ..............
+ 2 .....................................
+ 1. .....................................
+ .........................................
+
+
```

As can be seen after "1", the three Euler angles are equal to zero, since the polarization direction and the global OX axis are identical. Moreover, the value of the thickness after "2" is dummy, since the model is axisymmetrical.

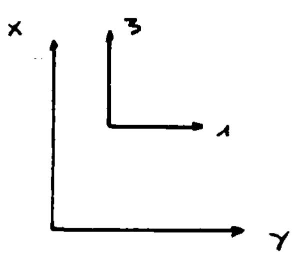
FIGURE 1. "CARTESIAN" POLARIZATION.

a/ definition of the Euler angles.

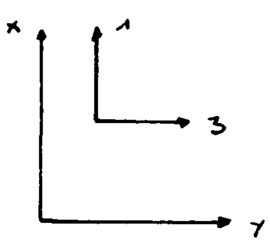


b/ examples.

Axisymmetrical case.



$\alpha = \beta = \gamma = 0$

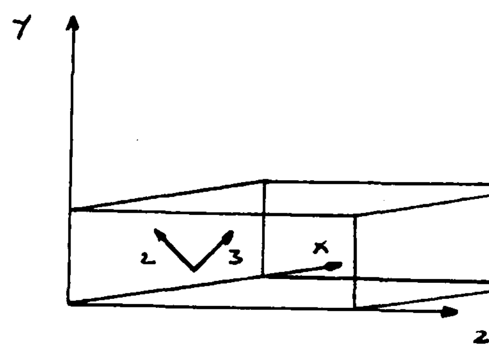


$\alpha = -90.$

$\beta = 0.$

$\gamma = 180.$

Three-dimensional case.



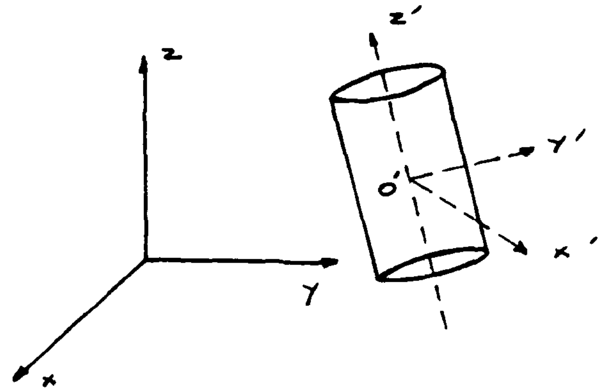
$\alpha = 90.$

$\beta = -45.$

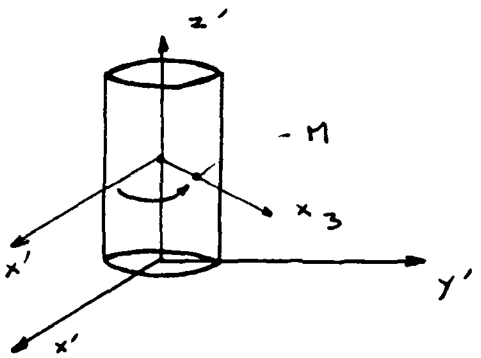
$\gamma = 180.$

FIGURE 2. "CYLINDRICAL" POLARIZATION.

a/ definition of the geometry.



b/ definition of the rotation processed by the code.



c/ definition of the DELTA angle.

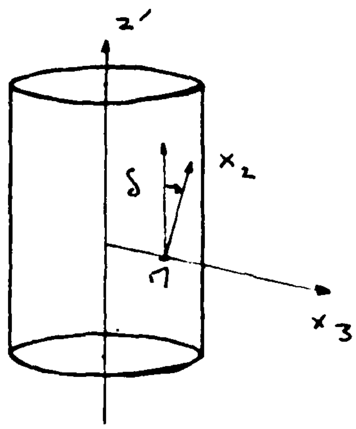
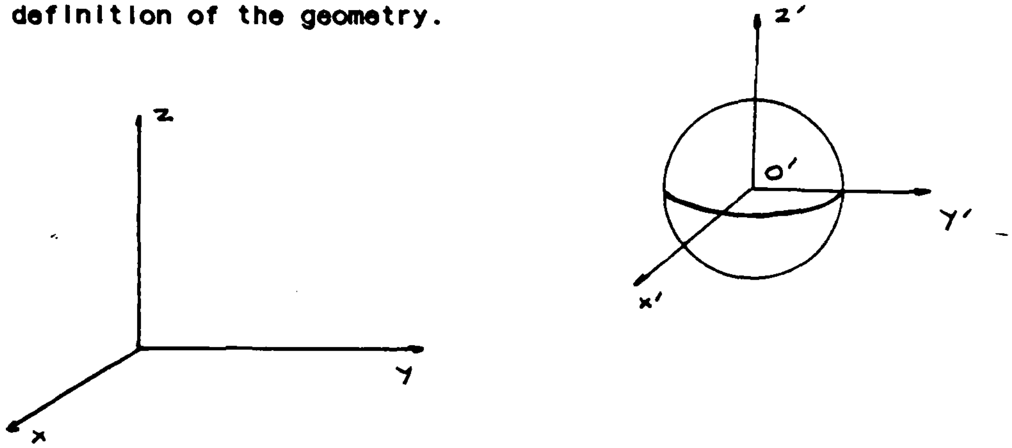
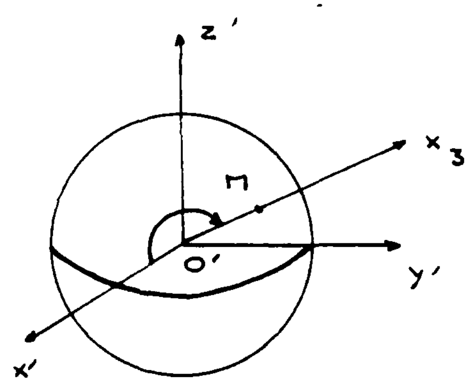


FIGURE 3. "SPHERICAL" POLARIZATION.

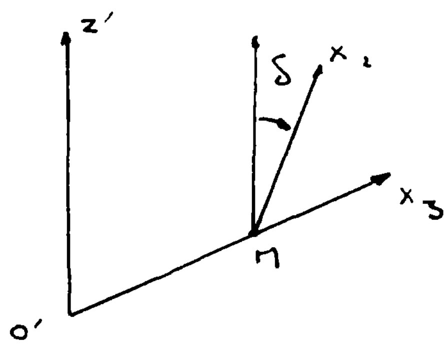
a/ definition of the geometry.



b/ definition of the rotations processed by the code.



c/ definition of the DELTA angle.



REDUCTIONS
I N MDF

This entry defines the frontal reduction when master degrees of freedom are requested (modal and harmonic analysis). In this case, it is compulsory.

. I is a control integer. If **I=0**, backsubstitution is processed to provide NLO full eigenvectors from the eigenmode number **N**. If **I=1**, backsubstitution is processed to provide NLO full eigenvectors the eigenfrequencies of which are above **N** (in this case, N is an integer value of a threshold frequency, given in Hz). The control integer N is mainly devoted to the elimination of the rigid body modes, which are provided by the eigenmode computation, from the backsubstitution. In some cases, it can be used to restrict the backsubstitution to a specific given mode. **Attention has to be paid** to the fact that only the twenty first eigenvectors can be backsubstituted, including the rigid body modes.

. **MDF** is the chosen number of master degrees of freedom which are automatically selected by the code. **Attention has to be paid** to the fact that electrical potential degrees of freedom are always condensed, except the applied potential in the case of an harmonic analysis which is a master degree forced by the code. Moreover, manually selected degrees of freedom can be defined in the set of data which specifies the boundary conditions. Finally, the degrees of freedom corresponding to damping elements are all master degrees and are forced by the code. Then, the sum of the total master degree number IDF (automatic masters + applied potential degree + degrees from damping elements) and the automatically reduced electrical degree number must be lower than the total degree of freedom number IDT.

Test example.

```
+
+ REDUCTIONS ...............
+ 0 1 23 ...................
+
+
```

Thus, all the available degrees of freedom are master degrees and, due to the fact that no rigid body mode exists in this case, the five first full eigenvectors will be provided.

FREQUENCIES
F1 F2 F3 ....FN

This entry is only requested for an harmonic analysis but then is compulsory. Its parameters are the values of the frequency for which a computation is needed. The maximum number of these values is 20 if there is no damping (in-air harmonic analysis) and 10 if damping is taken into account (in-water harmonic analysis). The parameter line can be extended using the character &.

Test example.

There is no test example for this entry since this entry is only concerned with an harmonic analysis.

RADIATION MONOPOLAR or **DIPOLAR**

This entry is only requested for the modelling of a radiating structure but then is compulsory.

Test example.

There is no test example for this entry since this entry is only concerned with an harmonic analysis of a radiating structure.

PRINTING
NPR

This entry defines the printing level **NPR**, which must equal to 1, 2 or 3, and corresponds to the amount of information available on the result listing. If **NPR=1**, only the computation results are printed. If **NPR=2**, the more important data are recalled and printed together with the computation results. This level is the normal (**default**) level. **NPR=3** corresponds to the trace level and is only used to detect faults.

Test example.

For this example, the default case is the best choice.

- **END**

This entry marks the end of the entry list and is compulsory.

Test example.

```
+
+ END .......................
+
+
```

3.4 THE LOADING AND BOUNDARY CONDITION DATA

The data which are associated with the loading and boundary conditions and which are described hereafter have to be written in fixed formats. Work is in progress to provide these conditions using specific entries in the next **ATILA** version. In all the data lines, a given **degree of freedom** is marked by the number **D**, with the following correspondence: 1 for UX (or P), 2 for UY, 3 for UZ, 4 for NX (or V), 5 for NY and 6 for NZ. Moreover, **lines** which are parallel to a global coordinate axis and **planes** which are normal to a global coordinate axis are also marked by a number, denoted **P**. Then, the correspondence is: 1 for a plane normal to the OX axis, 2 for a plane normal to the OY axis, 3 for a plane normal to the OZ axis, 4 for a line parallel to the OX axis, 5 for a line parallel to the OY axis and 6 for a line parallel to the OZ axis.

3.4.1 DESCRIPTION OF THE LOADING CONDITION DATA.

In the case of a static analysis, the lumped applied forces (or moments) have to be described using one of the two following procedures. The procedure is selected using a control number, named **CONT**, which has to be written first, in F10.3 format:

```
......CONT.........................................
```

Then, if **CONT=7777.0**, the nodes where the forces (or moments) are applied are defined by their **coordinates** and, for each applied force (or moment), the following line has to be written (3F10.3, 2I5, F10.3):

```
.........X.........Y.........Z....D..NLO.........F.....
```

X, Y, Z are the coordinates of the fulcrum, D is the application direction (1 to 3 for an applied force, 4 to 6 for an applied moment, following the above mentioned correspondence rules), NLO is the loading case number (from 1 to the value defined previously by the **NLOAD** entry), F is the force (or moment) value.

If **CONT=8888.0**, the nodes where the forces (or moments) are applied are defined by their **number** and, for each applied force (or moment), the following line has to be issued (3I5, F10.3):

```
.NODE....D..NLO.........F.............................
```

NODE is the application node number, D, NLO and F have the same meanings as in the previous case.
In each case, the set of loading condition data lines has to be closed by a dummy line.

WARNING

> **If the model is axisymmetrical, the force (or moment) actual value has to be divided by 2*PI.**
> **In the case of a distributed loading, the lumped nodal forces (or moments) have to be computed using the corresponding element shape functions.**

Test example.

The test example is a modal analysis and does not require any loading condition.

3.4.2 DESCRIPTION OF THE BOUNDARY CONDITION DATA.

The boundary conditions are generally associated with mechanical clampings, symmetry planes, electrodes, pressure released surfaces... They are defined by specific data lines, the classical format of which is (3I5, 4X, A1). The available cases are arranged in two groups. The first group contains only boundary conditions which refer to the global coordinate axes. The second group contains boundary conditions which require specific local coordinates.

Boundary conditions defined in global axes.

* The degree of freedom D is deleted at the node N.

```
....N....D.............................................
```

* The degree of freedom D is deleted for all the nodes which belong to a specific plane or line P containing the node N.

```
...-N....D....P....................................
```

This condition can be restricted to the solid or the fluid part of a mesh in the case of coupled problems by specifying S or F in the fourth field and using one of the two following lines:

```
...-N....D....P....S...............................
```

```
...-N....D....P....F...............................
```

* The degree of freedom D is deleted for all the nodes in the mesh.

```
.........D.............................................
```

* Degrees of freedom of type D are identical for all the nodes which belong to a specific line or plane P containing the node N.

...-N....D...-P...

* The degree of freedom D at node N is a master degree of freedom.

....N...-D..

* Degrees of freedom of type D are master degrees of freedom for all the nodes which belong to a specific line or plane P containing the node N.

...-N...-D....P...

* Degrees of freedom of type D are master degrees of freedom and identical for all the nodes which belong to a specific line or plane P containing the node N.

...-N...-D...-P...

* In an harmonic analysis of a piezoelectric structure, the degree of freedom associated with the **applied potential** has to be a master degree of freedom and specifically marked, since its numerical processing requires a special storage. This selection is realised using the following data line:

....N...-4.........1....................................

where N is a node which belongs to the related electrode. Moreover, this marked degree of freedom is the same for the whole electrode. Thus, if this electrode is a specific line or plane P, the data line can be written:

...-N...-4...-P....1....................................

* In an harmonic analysis of a piezoelectric structure using the **mixed plane wave-finite element method**, the degree of freedom associated with the displacement D normal to the interface between the two domains has to be a master degree of freedom and specifically marked, since its numerical processing requires a special storage. This selection is realised using the following data line:

....N...-D.........I....................................

where N is the node which corresponds to D. Moreover, this marked degree of freedom is the same for the whole interface. Thus, if this interface is a specific line or plane P, the data line can be written:

...-N...-D...-P....I....................................

* The degree of freedom of type D at node N is identical to the degree of freedom of the same type D at node M.

....N....D....M.......................................

Boundary conditions defined in local axes.

Local axes can be requested in two different cases. Firstly, the displacement which is constrained can be in a direction which is not parallel to the global axis (figure 1). Secondly, a boundary condition has to be applied on a line which is not parallel to a global axis or on a plane which is not perpendicular to a global axis (figure 2). In fact, these two cases are generally merged (figure 3).

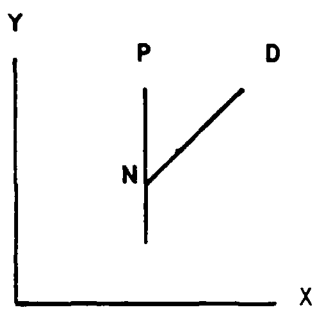


Figure 1.

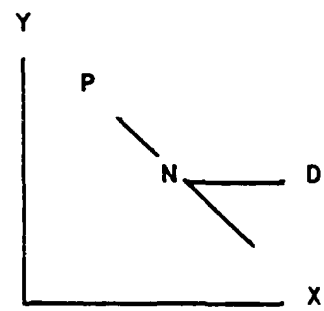


Figure 2.

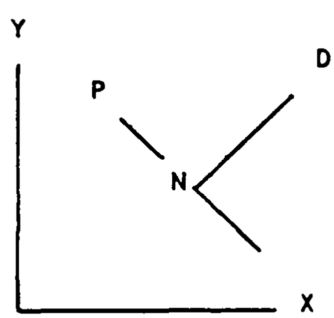


Figure 3.

* To define local axes at a given node N, the user has to issue the following lines (formats 2I5 and 6F10.3).

```
....N...10.........................................

.......A11.......A12.......A13.......A21.......A22..../
                                                  ...A23
```

A11, A12, A13 are the direction cosines of the first local axis and A21, A22, A23 those of the second local axis. Then, degrees of freedom at node N are automatically defined by the code in the new local axes and they can be constrained using the same data lines as for the first group. If the same global axes have to be defined for all the nodes which belong to the same line or plane P containing the node N, the preceeding data set has to be modified by simply substituting for the first line the following line:

```
...-N...10....P.....................................
```

* To constrain nodes belonging to a plane which is not perpendicular to a global axis, the corresponding line of the first group has to be modified by substituting 9001 to P and adding immediately after a second line which contains the direction cosines A1, A2 and A3 of the normal to this new plane (format 3F10.3). Thus, if the degrees of freedom of type D of all the nodes which belong to this plane have to be deleted, the two data lines are:

```
...-N....D.9001.....................................

........A1........A2........A3.........................
```

To constrain nodes belonging to a line which is not parallel to a global axis, the corresponding line of the first group has to be modified by substituting 9004 to P and adding immediately after a second line which contains the direction cosines A1, A2 and A3 of this new line (format 3F10.3). The preceeding example can be immediatly transposed.

* Finally, the two preceeding cases can be merged simply by mixing the corresponding data lines. Thus, if the preceeding local axes have to be defined for all the nodes belonging to the preceeding plane, the following data lines have to be issued:

```
...-N...10.9001.....................................

.......A11.......A12.......A13.......A21.......A22..../
                                                  ...A23

........A1........A2........A3.........................
```

General comments.

. This data line set has to be closed by a dummy line.
. Constraints on different degrees of freedom can be merged on the same data line by placing side by side the corresponding integers in the definition of the D integer. Thus, if the same constraint has to be applied to UX, UY and NZ, D has to be equal to 126.
. Null flux condition is the natural condition of the finite element formulation used in ATILA and is implicit. This is the case for the fluid velocity in fluid domains and for the electric field in piezoelectric domains.

Test example.

```
+
+.........3..................
+...-1...14....5.............
+...-6....4....5.............
+............................
+
+
```

The first line deletes the UZ component of the displacement field (compulsory, due to the fact than LCPDDC=7). The second line is the symmetry condition along the OY axis. The third line is the open electrode condition on the upper side of the ceramic ring.

3.5 THE MIXED MODEL SPECIFIC DATA

When a mixed finite element-plane wave model is used, the finite element part is described simply following the guide lines given in the other sections. This section is devoted to the description of the plane wave part of the structure, which is assumed to be organised as shown below.

```
                          :                       :               :
. . . . . . . . . .       :  . . . . . . . . . . .:  . . . . . .  :
.////////////.            :  .                    :               :
.////////////.            :  .      PART 1        :  .            :
.////////////.            :  .                    :               :
./ FINITE //.             :  . . . . . . . . . . .:               :
./ ELEMENT /.             :                       :     PART 3    :
./ DOMAIN //.             :  . . . . . . . . . . .:               :
.////////////             :  .                    :               :
 ////////////.            :  .      PART 2        :  .            :
.////////////.            :  .                    :               :
                          :                       :
                          P1                      P2              P3
```

The plane wave part is limited on one side by its interface **P1** with the finite element domain. This interface has to be a plane and its normal displacement is assumed to be uniform. It is limited on the other side by another plane face **P3**, the normal displacement of which is free or clamped. Finally, it can be composed of one, two or three parts. **Part 1** is the active part and its presence is compulsory. **Part 3** is passive and in series with part 1. It can be missing. **Part 2** is also passive and in parallel with part 1. It can be missing, but its presence requires the presence of part 3. In the classical case of Tonpilz transducers, part 1 is the ceramic stack, part 2 the prestress rod and part 3 the tail mass, but other schemes can be described if they follow the previous general organisation. The description of the structure is made in the compulsory order: part 1, part 2, part 3. All the constitutive elements are assumed to [illegible] conical shape.

The data lines are the [illegible] named IND [illegible]

IND

If **IND=1**, the structure is axisymmetrical. If **IND=2**, the structure is three-dimensional with one symmetry plane. If **IND=3**, the structure is three-dimensional with two symmetry planes. The following lines describe the successive slices of part 1, 2 and 3. Their format is (2I5, 6E10.3). They have to be written under the form:

```
..ITO..ITH........TH.......OR1.......IR1.......OR2..../
                                        ...IR2.......PHI
```

where:

ITO = **NM**, where N and M are two integers defined hereafter,
ITH = 0 or 180 and allows to reverse the ceramic polarization direction. Its value is dummy for other materials,
TH is the thickness of the slice,
OR1 is the outside radius of the slice first face,
IR1 is the inside radius of the slice first face,
OR2 is the outside radius of the slice second face,
IR2 is the inside radius of the slice second face,
PHI is the applied potential (beware of the sign), if any.

N defines the type of the slice

N = 1 for an elastic cylinder (part 1 or 3),
= 2 for a piezoelectric ceramic cylinder (part 1),
= 3 for an elastic cone (part 1 or 3),
= -1, -2 or -3 if this slice is the last slice in part 3,
= 4 for an elastic cylinder (part 2),
= 5 for a conical cylinder (part 2),
= -4 or -5 if this slice is the last slice in part 2.

M defines the slice material **

M = 1 for 25CD4 steel or X5105 ceramic,
= 2 for AU4G or X9 ceramic,
= 3 for brass (UZ39PB2) or X31 ceramic,
= 4 for [illegible] steel ([illegible]) or X37 ceramic.

To use this set, the user has to write down the following data line 5

[illegible] IP3

If IP3 is equal to zero, the normal displacement to P3 is free, while if IP3 is equal to 1 this normal displacement is clamped.

WARNING: With this version, only 39 successive slices can be accomodated by the code.

---

*. Cylindrical elements can be replaced by elements with different cross section shapes (squares, rectangles ...), if the plane wave assumption is still correct in these cases. Then, the inside and outside radius of the corresponding cylindrical slices, as defined in page 34, have to be computed by the user to ensure that the cross section area values are the actual values.

**. Material property values are provided by two DATA instructions in a subroutine named TAC. The user can modify them by editing the corresponding FORTRAN file. For elastic materials, values are listed in the DATA/SM/ instruction, with two values for each material (density and Young's modulus), successive materials being put in the M ascending order. For piezoelectric materials, values are listed in the DATA/SP/ instruction, with eleven values for each material (density, constant electric field elastic constants S_{33}, S_{13}, S_{11}, S_{12}, S_{44}, piezoelectric constants d_{33}, d_{31}, d_{15} and constant strain dielectric constants EP_{33} AND EP_{11}). Successive materials are also put in the M ascending order.

3.6 THE GRAPHIC DISPLAY SPECIFIC DATA

With the **ATILA** code, graphic displays can be obtained using different devices, mainly plotters and workstations. At the present time, results have been obtained with BENSON plotting tables, VERSATEC Xerox tables, Tektronix 4014, 4114, 4115, as well as MASSCOMP and MICROVAX computers. When plotters have to be used, the **ATILA** run generates a file which contains the required information for the plotter and this plotter is generally activated in a following job. In this case, graphic display specific data have to be added to the data file and are described hereafter. When a workstation is used, the graphic part of the job is interactive and no data which are related to the graphic display have to be put in the data file. Nevertheless, during the run of the interactive job, questions are submitted to the user which correspond strictly to the following information.

Graphic displays of the structure in its rest position or in deformed shapes can be easily obtained using the following set of data. For each graphic display, two data lines are requested with the respective formats (4F10.3, 3I5) and (A40, F10.3). These lines are:

.....ALPHA......BETA......XLTH......BRTH...NP..NE1..NE2

...................................TITLE.....SCALE.....

* ALPHA and BETA are angles (degrees) which define the projection of the structure on the drawing plane. The structure being initially observed by the user in the OY direction, out of this user, the OX axis is rotated of the ALPHA angle around the OZ axis. Then, the initial OZ axis is rotated of the BETA angle around the original OX axis. Finally, the structure is projected in the initial OXZ plane. ALPHA and BETA are positive if the axis are moving back of the initial OXZ plane.

* XLTH and BRTH (in meters) define the space occupied by the graphic display. If BRTH equals zero and XLTH is negative, -XLTH is the scale of the display with respect to the actual size of the structure.

* NP defines the type of display, with the following correspondences:

NP = 0 . Exploded view of the mesh.
NP = 2 . Exploded view of the mesh, with the node numbering.
NP =-2 . Exploded view of the mesh, with the element numbering.

NP = 3 . Dashed view of the mesh, with the degrees of freedom numbering.
NP = 7 . Simple display of the mesh (full lines).
NP =12 . Display of the mesh with full outside lines and dashed inside lines.
NP =31 tO 49 . Graphic display of the deformed shape of the structure, in the loading case, full eigenvector or excitation frequency order. The deformed shape is in full line while the rest position is in dashed line. If damping is taken into account in the computation, two graphic displays have to be issued for each frequency (real and imaginary part of the displacement field).

For NP = 0, 2, -2, 3, 7 and 12 the display is provided by a preprocessor. If NP = 31 to 49, the display is provided by a postprocessor. Several displays can be simultaneously requested in each of these two categories.

* NE1 and NE2 permit getting the graphic display of a part of the structure only. In this case, NE1 is the number of the first drawn element and NE2 the number of the last. If these integers are equal to zero, the whole stucture is displayed. Care must be given to the case of fluid domain, since in this case the finite element variable is the pressure which cannot be represented as a displacement. So, fluid elements have to be excluded from graphic displays.

* A specific TITLE can be given to each graphic display This title cannot contain more than 40 letters.

* SCALE is a multiplying factor for the displacement amplitude. It allows adjustment of this amplitude value for sake of clarity. In the case of an eigenmode computation, for which the sign of the whole displacement field is not absolutely defined, it allows also to reversal of this sign for the sake of comparison between results of different computations (in the case of piece part analysis, for example).

```
 AAAAAAAA    TTTTTTTTTT    II   LL          AAAAAAAA
AA      AA       TT        II   LL         AA      AA
AA      AA       TT        II   LL         AA      AA
AAAAAAAAAA       TT        II   LL         AAAAAAAAAA
AA      AA       TT        II   LL         AA      AA
AA      AA       TT        II   LL         AA      AA
AA      AA       TT        II   LLLLLLL    AA      AA
```

FINITE ELEMENT MODELING

OF

PIEZOELECTRIC TRANSDUCERS

AUGUST 1987

CHAPTER 4
ELEMENT LIBRARY

THE FINITE ELEMENT CODE ATILA HAS BEEN [illegible] BY THE A[illegible] LABORATORY OF THE INSTITUT SUPERIEUR D ELECTRONIQUE DU NORD ([illegible] FRANCE) FOR THE GROUPE D ETUDES ET DE RECHERCHE EN DETECTION SOUS-MARINE (GERDSM [illegible] FRENCH NAVY [illegible]) [illegible] SOFTWARE ENGINEERING [illegible] BY THE SINAPTEC COMPANY [illegible] FRANCE

4.1 INTRODUCTION

This chapter describes the finite elements which are available in the **ATILA** code*. For each element, it provides its present name, its name in the previous **ATILA** version (in parenthesis), its definition, the list of the degrees of freedom which are attached to its nodes (translations, rotations, electrical potential, pressure), its entry parameters (topology, material properties, geometrical properties).

Most of the elements in the **ATILA** library are **isoparametric**. Thus, complex structures with curved sides or faces can be easily modelled. Nevertheless, **best results are obtained when these elements have reasonable shapes**. Wild distortions lead to inaccuracy because the assumptions made within the code for strains become unrealistic. Extreme distortion may even cause program failure. The user should guard against excessive distortion by adhering to the following guide lines for 2D elements or for faces of 3D elements:

- the angles between adjacent sides of quadrilaterals should be between 45 and 135 degrees,
- the angles between adjacent sides of triangles should be between 30 and 100 degrees,
- in elements with curved sides, the radius of curvature of each side should exceed the length of the longest side.

Warning: all data must be given using **MKS** units or in any coherent unit system which is deduced from the MKS system.

* [illegible] test [illegible] several [illegible] elements are in progress [illegible] A [illegible] request.

4.2 ATILA ELEMENT DIRECTORY

NATURE	GEOM	TYPE	N	NAME	COMMENT	P
ELASTIC	HEXA	3D	20	HEXA20E		4
	PRIS	3D	15	PRIS15E		5
	QUAD	2D&AXI	8	QUAD08E		6
	QUAD	PLATE	8	PLAT08E		7
	QUAD	SHELL	8	FACE08E	FACET	8
	TRIA	2D&AXI	6	TRIA06E		9
	TRIA	PLATE	6	PLAT06E		10
	LINE	SHELL	3	SHEL03E	AXI	11
		AXI	5	TRAN05E	TRANS	12
	LINE	SPRING	2	SPRI02E		13
PIEZO.	HEXA	3D	20	HEXA20P		14
	PRIS	3D	15	PRIS15P		15
	QUAD	AXI	8	AXIS08P		16
	QUAD	2D	8	QUAD08P	PL.ST	17
	TRIA	AXI	6	AXIS06P		18
	TRIA	2D	6	TRIA06P	PL.ST	19
FLUID	HEXA	3D	20	HEXA20F		20
	PRIS	3D	15	PRIS15F		21
	QUAD	AXI	8	AXIS08F		22
	TRIA	AXI	6	AXIS06F		23
SPECIAL	QUAD	3D	16	QUAD16I	INTER	24
	TRIA	3D	12	TRIA12I	INTER	25
	LINE	AXI	6	AXIS06I	INTER	26
	QUAD	3D	8	QUAD08M	MONOP	27
	TRIA	3D	6	TRIA06M	MONOP	28
	LINE	AXI	3	AXIS03M	MONOP	29
	LINE	AXI	3	AXIS03D	DIPOL	30

GEOM: geometrical shape of the element.
N number of nodes.
NAME: name of the element, as required for the entry.
P page number.

HEXA : hexaedra.
PRIS : triangular prism.
QUAD : quadrilateral.
TRIA : triangle.
INTER: interface element.
MONOP: monopolar damper.
DIPOL: dipolar damper.
PL.ST: plane strain.
TRANS: transition element.

4.3 ELEMENT DESCRIPTION

Descriptions of a variety of elements are provided on the following pages.

HEXA20E (R37110).

DESCRIPTION.

.HEXA20E is a twenty node isoparametric hexaedra, used for homogeneous isotropic elastic materials.

.The degrees of freedom are, for each node: u_x, u_y, u_z (3 translations).

TOPOLOGY.

```
                                 7 . . . .20 . . . . 8
                                 . .                 . .
                                . .                 . .
                             18 .                19 .
                              . 15               . 16
                             .   .              .   .
                           5 . . . .17 . . . . 6    .
                           .     .             .    .
                           .     3 . . . .12 . . . . 4
                           .    .              .   .
      Z                   13  .              14  .
      .  Y                 . 10              . 11
      . .                  . .               . .
      ..                   ..                ..
      0 . . . X            1 . . . . 9 . . . . 2
```

ENTRIES.

."ELEMENTS" entry parameters.
HEXA20E MATER
N1 N2 N3 N4 N5 N6 N7 N8 N9 N10 N11 N12 N13 N14 N15 N16 N17 N18 N19 N20
where N1,...N20 are the actual node numbers corresponding to the integers 1,...20 on the above figure.

."MATERIALS" entry parameters.
E NU RO
where E is Young's modulus (Pa), NU is Poisson's ratio and RO is the density (kg/m^3).

."GEOMETRY" entry parameters.
No.

COMMENTS.

.Generally, the aspect ratio has to be less than 3.

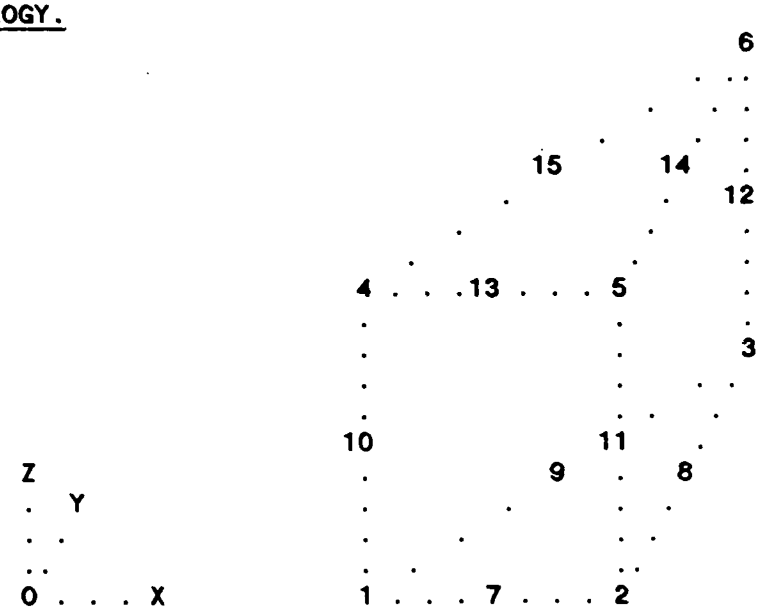
PRIS15E (R37210).

DESCRIPTION.

.PRIS15E is a fifteen node isoparametric triangular prism, used for homogeneous isotropic elastic material.

.The degrees of freedom are, for each node: u_x, u_y, u_z (3 translations).

TOPOLOGY.



ENTRIES.

."ELEMENTS" entry parameters.
PRIS15E MATER
N1 N2 N3 N4 N5 N6 N7 N8 N9 N10 N11 N12 N13 N14 N15
where N1,...N15 are the actual node numbers corresponding to the integers 1,...15 on the above figure.

."MATERIALS" entry parameters.
E NU RO
where E is Young's modulus (Pa), NU is Poisson's ratio and RO is the density (kg/m^3).

."GEOMETRY" entry parameters.
No.

COMMENTS.

.Generally, the aspect ratio has to be less than 3

QUAD08E (R36210).

DESCRIPTION.

.QUAD08E is an eight node isoparametric quadrilateral, used for homogeneous isotropic elastic material and for plane stress, plane strain or axisymmetrical analysis.

.The degrees of freedom are, for each node: u_x, u_y (2 translations).

TOPOLOGY.

```
                    3 . . . . 8 . . . . 4
                    .                   .
                    .                   .
                    .                   .
                    .                   .
                    6                   7
  Y                 .                   .
  .                 .                   .
  .                 .                   .
  .                 .                   .
  0 . . . X         1 . . . . 5 . . . . 2
```

ENTRIES.

."ELEMENTS" entry parameters.
QUAD08E MATER NG
N1 N2 N3 N4 N5 N6 N7 N8
where N1,...N8 are the actual node numbers corresponding to the integers 1,...8 on the above figure.

."MATERIALS" entry parameters.
E NU RO
where E is Young's modulus (Pa), NU is Poisson's ratio and RO is the density (kg/m^3).

."GEOMETRY" entry parameter.
T
where T is the thickness of the element (ignored in the case of plane strain and axisymmetrical analyses).

COMMENTS.

.Generally, the aspect ratio has to be less than 3.

PLATO8E (R38110).

DESCRIPTION.

.PLATO8E is an eight node flat quadrilateral plate element, used for homogeneous isotropic elastic material. It relies on the classical Love-Kirchhoff hypothesis.

.The degrees of freedom are, for each node: u_z, n_x, n_y (1 translation, 2 rotations).

TOPOLOGY.

```
                     3 . . . . 8 . . . . 4
                     .                   .
                     .                   .
                     .                   .
                     .                   .
                     6                   7
       Y             .                   .
       .             .                   .
       .             .                   .
       .             .                   .
       O . . . X     1 . . . . 5 . . . . 2
```

ENTRIES.

."ELEMENTS" entry parameters.

PLATO8E MATER NG

N1 N2 N3 N4 N5 N6 N7 N8

where N1, ..., N8 are the actual node numbers corresponding to the integers 1, ..., 8 on the above figure.

."MATERIALS" entry parameters.

E NU RO

where E is Young's modulus (Pa), NU is Poisson's ratio and RO is the density (kg/m^3).

."GEOMETRY" entry parameter.

T

where T is the thickness of the element.

COMMENTS.

.Generally, the aspect ratio has to be less than 3.
.The longitudinal and lateral dimensions of the whole plate have to be at least ten times the thickness.
.This element can be superimposed to the QUAD08E element (plane stress condition) to generate a flat shell element (facet).

FACE08E (R44210).

DESCRIPTION.

.FACE08E is an eight node quadrilateral shell element, used for homogeneous isotropic elastic material. It relies on the classical Love-Kirchhoff hypothesis and has to be flat (facet).

.The degrees of freedom are, for each node: u_x, u_y, u_z, n_x, n_y (3 translations, 2 rotations).

TOPOLOGY.

```
                    3 . . . . 8 . . . . 4
                    .                   .
                    .                   .
                    .                   .
                    .                   .
                    6                   7
        Y           .                   .
        .           .                   .
        .           .                   .
        .           .                   .
        0 . . . X   1 . . . . 5 . . . . 2
```

ENTRIES.

."ELEMENTS" entry parameters.
FACE08E MATER NG
N1 N2 N3 N4 N5 N6 N7 N8
where N1,...N8 are the actual node numbers corresponding to the integers 1,...8 on the above figure.

."MATERIALS" entry parameters.
E NU RO
where E is Young's modulus (Pa), NU is Poisson's ratio and RO is the density (kg/m^3).

."GEOMETRY" entry parameter.
[illegible]
[illegible] is the thickness of the element .

COMMENTS.

[illegible] the aspect ratio has to be less than 3.
[illegible] and lateral dimensions of the whole shell have [illegible] least ten times the thickness.
[illegible] is equivalent to the superposition of the QUAD08E [illegible] condition) and PLAT08E element.

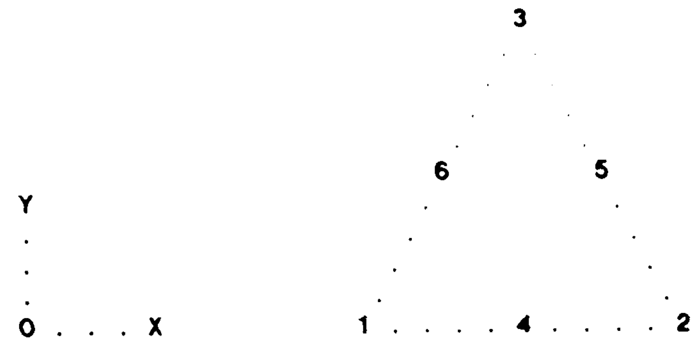
TRIAO6E

DESCRIPTION.

.TRIAO6E is a six node isoparametric triangle used for homogeneous isotropic elastic material and for plane stress, plane strain or axisymmetrical analysis.

.The degrees of freedom are, for each node u_x, u_y (2 translations).

TOPOLOGY.



ENTRIES.

."ELEMENTS" entry parameters.
TRIAO6E MATER NG
N1 N2 N3 N4 N5 N6
where N1,...N6 are the actual node numbers corresponding to the integers 1,...6 on the above figure.

."MATERIALS" entry parameters.
E NU RO
where E is Young's modulus (Pa), NU is Poisson's ratio and RO is the density (kg/m^3).

."GEOMETRY" entry parameter.
T
where T is the thickness of the element (ignored in the case of plane strain and axisymmetrical analyses).

COMMENTS.

.Generally, the aspect ratio has to be less than 3.

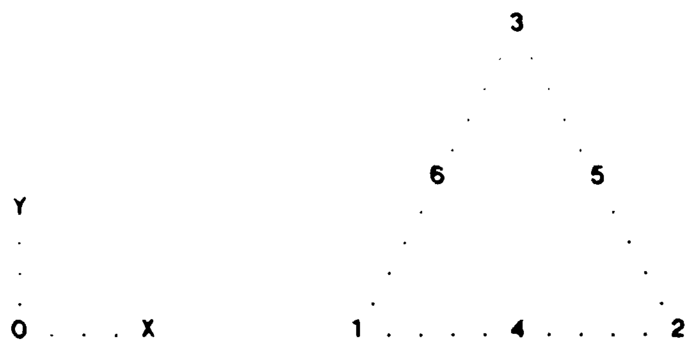
PLATO6E R384

DESCRIPTION

PLATO6E is a six node flat triangular plate element, used for homogeneous isotropic elastic material. It relies on the classical Love Kirchhoff hypothesis.

The degrees of freedom are, for each node: u_z, n_x, n_y (translation, 2 rotations).

TOPOLOGY



ENTRIES.

."ELEMENTS" entry parameters.
PLATO6E MATER NG
N1 N2 N3 N4 N5 N6
where N1,...N6 are the actual node numbers corresponding to the integers 1,...6 on the above figure.

."MATERIALS" entry parameters.
E NU RO
where E is Young's modulus (Pa), NU is Poisson's ratio and RO is the density (kg/m^3).

."GEOMETRY" entry parameter.
T
where T is the thickness of the element.

COMMENTS.

.Generally, the aspect ratio has to be less than 3.
.The longitudinal and lateral dimensions of the whole plate have to be at least ten times the thickness.
.This element can be superimposed to the TRIA06E element (plane stress condition) to generate a flat shell element (facet).

SHEL03E (R42123).

DESCRIPTION.

.SHEL03E is a three node axisymmetrical thin shell element, the formulation of which takes account of its double curvature. It relies on the classical Love-Kirchhoff hypothesis.

.The degrees of freedom are, for each node: u_x, u_y, n_z (2 translations, 1 rotation).

TOPOLOGY.

```
Y
.
.
.
O . . . X          1 . . . . 2 . . . . 3
```

ENTRIES.

."ELEMENTS"entry parameters.
SHEL03E MATER NG
N1 N2 N3
where N1, N2, N3 are the actual node numbers corresponding to the integers 1, 2, 3 on the above figure.

."MATERIALS" entry parameters.
E NU RO
where E is Young's modulus (Pa), NU is Poisson's ratio and RO is the density (kg/m^3).

."GEOMETRY" entry parameter.
T R
where T is the thickness of the element and R its radius of curvature (around the axis).

COMMENTS.

.The longitudinal and radial dimensions of the whole shell have to be at least ten times the thickness.

TRANO5E (R43000).

DESCRIPTION.

.TRANO5E is a five node axisymmetrical element which ensures the transition between quadrilateral or triangular axisymmetrical elements and the previous thin shell axisymmetrical element. It allows an accurate description of their interaction. It includes a solid part (nodes 1, 2, and 3) and a shell part (nodes 4 and 5).

.The degrees of freedom are, for nodes 1, 2 and 3: u_x, u_y (2 translations), and for nodes 4 and 5: u_x, u_y, n_z (2 translations, 1 rotation).

.

TOPOLOGY.

```
Y             1-------*-------*     1
.             .               !     .
.             .               !     .
.             2 . . . 4 . . . 5     2 . . . 4 . . . 5
.             .               !     .
.             .               !     .
O . . . X     3-------*-------*     3
```

ENTRIES.

."ELEMENTS"entry parameters.
TRANO5E MATER NG
N1 N2 N3 N4 N5
where N1, N2 , N3, N4, N5 are the actual node numbers corresponding to the integers 1, 2, 3, 4, 5 on the above figure.

."MATERIALS" entry parameters.
E NU RO
where E is Young's modulus (Pa), NU is Poisson's ratio and RO is the density (kg/m^3).

."GEOMETRY" entry parameter.
T R
where T is the thickness of the element and R its radius of curvature (around the axis).

COMMENTS.

.The longitudinal and radial dimensions of the whole shell have to be at least ten times the thickness.

SPRI02E (R30100).

DESCRIPTION.

.SPRI02E is a two node spring element, which is only able to transmit to its two limiting nodes a restoring force due to a change of its length. Its classical use is for the modelling of prestress bolts in transducer design.

.The degrees of freedom are, for each node u_x, u_y, u_z (3 translations).

TOPOLOGY.

O . . . X   1 2

ENTRIES.

."ELEMENTS" entry parameters.
SPRI02E MATER NG
N1 N2
where N1 and N2 are the actual node numbers corresponding to the integers 1 and 2 on the above figure.

."MATERIALS" entry parameters.
E .0 RO
where E is Young's modulus (Pa) and RO is the density (kg/m^3).

."GEOMETRY" entry parameter.
S
where S is the cross section area.

COMMENTS.

.In axisymmetrical models, the actual area S has to be divided by 2*pi. In three-dimensional models, if the rod belongs to N symmetry planes, its actual area S has to be divided by N+1.
.This element is nothing more than a classical spring. Attention has to be paid to spurious modes which can be due to the simple spring hypothesis.

HEXA20P (R3751)

DESCRIPTION.

.**HEXA20P** is a twenty node isoparametric hexaedra, designed for any piezoelectric material.

.The degrees of freedom are, for each node u_x, u_y, u_z, v (3 translations, 1 electrical potential).

TOPOLOGY.

```
                                7 . . . .20 . . . . 8
                               . .                 . .
                              . .                 . .
                           18 .                19 .
                            . 15                . 16
                           .                   .     .
                          5 . . . .17 . . . . 6     .
                          .     .             .     .
                          .     3 . . . .12 . . . . 4
                          .    .              .    .
  Z                      13 .              14 .
  .  Y                    . 10              . 11
  . .                     . .               . .
  ..                      ..                ..
  0 . . . X               1 . . . . 9 . . . . 2
```

ENTRIES.

."ELEMENTS" entry parameters.
HEXA20P MATER NG
N1 N2 N3 N4 N5 N6 N7 N8 N9 N10 N11 N12 N13 N14 N15 N16 N17 N18 N19 N20
where N1,...N20 are the actual node numbers corresponding to the integers 1,...20 on the above figure.

."MATERIALS" entry parameters.
The whole elastic,piezoelectric and dielectric tensors (see the detailed description of the entry: **MATERIALS**).

."GEOMETRY" entry parameters.
The new origin coordinates and the Euler angles which define the natural axes of the material (see the detailed description of the entry: **GEOMETRY POLARIZATION**).

COMMENTS.

.Generally, the aspect ratio has to be less than 3.
.The entry "**GEOMETRY**" has to be issued under the special form "**GEOMETRY POLARIZATION** (CARTESIAN, CYLINDRICAL, SPHERICAL)".

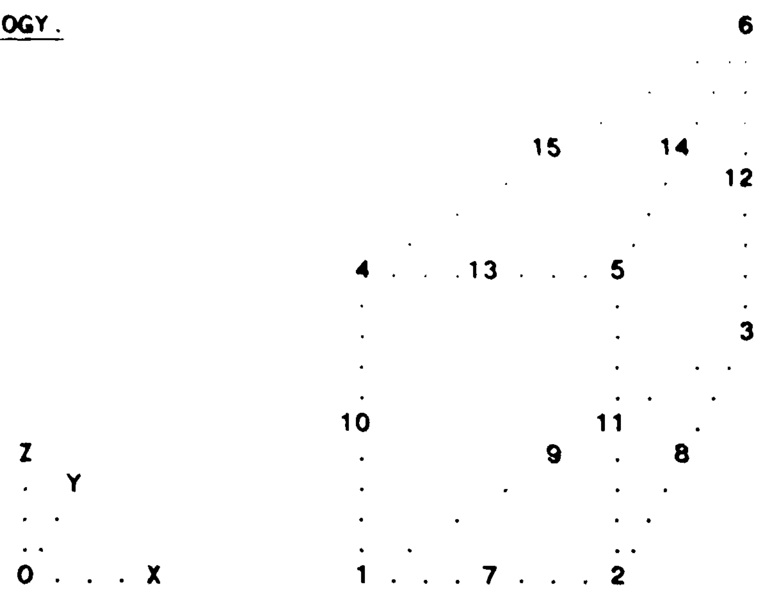
PRIS15P

DESCRIPTION.

PRIS15P is a fifteen node isoparametric triangular prism, designed for any piezoelectric material.

The degrees of freedom are, for each node u_x, u_y, u_z, V (3 translations, 1 electrical potential).

TOPOLOGY.



ENTRIES.

."ELEMENTS" entry parameters.
PRIS15P MATER NG
N1 N2 N3 N4 N5 N6 N7 N8 N9 N10 N11 N12 N13 N14 N15
where N1,...N15 are the actual node numbers corresponding to the integers 1,...15 on the above figure.

."MATERIALS" entry parameters.
The whole elastic,piezoelectric and dielectric tensors (see the detailed description of the entry: **MATERIALS**).

."GEOMETRY" entry parameters.
The new origin coordinates and the Euler angles which define the natural axes of the material (see the detailed description of the entry: **GEOMETRY POLARIZATION**).

COMMENTS.

.Generally, the aspect ratio has to be less than 3.
.The entry **"GEOMETRY"** has to be issued under the special form **"GEOMETRY POLARIZATION** (CARTESIAN, CYLINDRICAL, SPHERICAL)**"**.

AXISO8P (R37521).

DESCRIPTION.

.**AXISO8P** is an eight node isoparametric quadrilateral, designed for any piezoelectric material and used only for axisymmetrical analysis.

.The degrees of freedom are, for each node: u_x, u_y, v (2 translations, 1 electrical potential).

TOPOLOGY.

```
                        3 . . . . 8 . . . . 4
                        .                   .
                        .                   .
                        .                   .
                        .                   .
                        6                   7
     Y                  .                   .
     .                  .                   .
     .                  .                   .
     .                  .                   .
     0 . . . X          1 . . . . 5 . . . . 2
```

ENTRIES.

."ELEMENTS" entry parameters.
AXISO8P MATER NG
N1 N2 N3 N4 N5 N6 N7 N8
where N1,...N8 are the actual node numbers corresponding to the integers 1,...8 on the above figure.

."MATERIALS" entry parameters.
The whole elastic, piezoelectric and dielectric tensors (see the detailed description of the entry: **MATERIALS**).

."GEOMETRY" entry parameters.
The new origin coordinates and the Euler angles which define the natural axes of the material (see the detailed description of the entry: **GEOMETRY POLARIZATION**).

COMMENTS.

.Generally, the aspect ratio has to be less than 3.
.The entry **"GEOMETRY"** has to be issued under the special form **"GEOMETRY POLARIZATION (CARTESIAN, CYLINDRICAL, SPHERICAL)"**.

QUAD08P (R37501).

DESCRIPTION.

.**QUAD08P** is an eight node isoparametric quadrilateral, designed for any piezoelectric material and used only for plane strain analysis.

.The degrees of freedom are, for each node: u_x, u_y, v (2 translations, 1 electrical potential).

TOPOLOGY.

```
                         3 . . . . 8 . . . . 4
                         .                   .
                         .                   .
                         .                   .
                         .                   .
                         6                   7
       Y                 .                   .
       .                 .                   .
       .                 .                   .
       .                 .                   .
       0 . . . X         1 . . . . 5 . . . . 2
```

ENTRIES.

."ELEMENTS" entry parameters.
QUAD08P MATER NG
N1 N2 N3 N4 N5 N6 N7 N8
where N1,...N8 are the actual node numbers corresponding to the integers 1,...8 on the above figure.

."MATERIALS" entry parameters.
The whole elastic, piezoelectric and dielectric tensors (see the detailed description of the entry: **MATERIALS**).

."GEOMETRY" entry parameters.
The new origin coordinates and the Euler angles which define the natural axes of the material (see the detailed description of the entry: **GEOMETRY POLARIZATION**).

COMMENTS.

.Generally, the aspect ratio has to be less than 3.
.The entry "**GEOMETRY**" has to be issued under the special form "**GEOMETRY POLARIZATION** (CARTESIAN, CYLINDRICAL, SPHERICAL)".

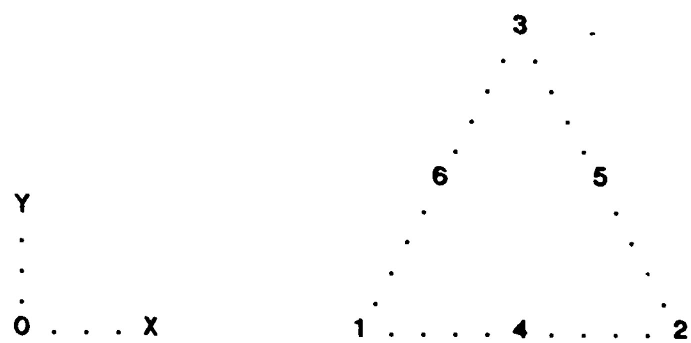
AXISO6P (R37522).

DESCRIPTION.

.**AXISO6P** is a six node isoparametric triangle, designed for any piezoelectric material and used only for axisymmetrical analysis.

.The degrees of freedom are, for each node: u_x, u_y, v (2 translations, 1 electrical potential).

TOPOLOGY.



ENTRIES.

."ELEMENTS" entry parameters.
AXISO6P MATER NG
N1 N2 N3 N4 N5 N6
where N1,...N6 are the actual node numbers corresponding to the integers 1,...6 on the above figure.

."MATERIALS" entry parameters.
The whole elastic, piezoelectric and dielectric tensors (see the detailed description of the entry: **MATERIALS**).

."GEOMETRY" entry parameters.
The new origin coordinates and the Euler angles which define the natural axes of the material (see the detailed description of the entry: **GEOMETRY POLARIZATION**).

COMMENTS.

.Generally, the aspect ratio has to be less than 3.
.The entry "**GEOMETRY**" has to be issued under the special form "**GEOMETRY POLARIZATION** (CARTESIAN, CYLINDRICAL, SPHERICAL)".

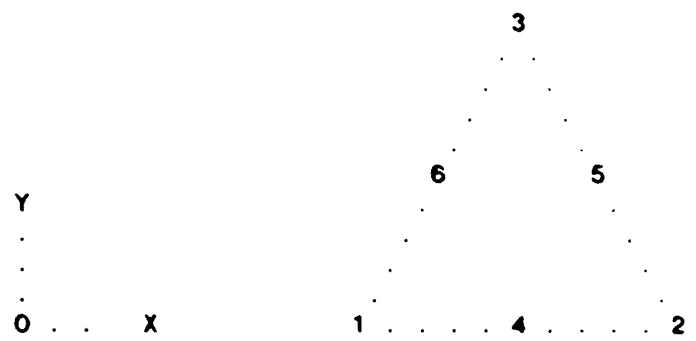
TRIA06P (R37502).

DESCRIPTION.

.**TRIA06P** is a six node isoparametric triangle, designed for any piezoelectric material and used only for plane strain analysis.

.The degrees of freedom are, for each node u_x, u_y, v (2 translations, 1 electrical potential).

TOPOLOGY.



ENTRIES.

."ELEMENTS" entry parameters.
TRIA06P MATER NG
N1 N2 N3 N4 N5 N6
where N1,...N6 are the actual node numbers corresponding to the integers 1,...6 on the above figure.

."MATERIALS" entry parameters.
The whole elastic, piezoelectric and dielectric tensors (see the detailed description of the entry: **MATERIALS**).

."GEOMETRY" entry parameters.
The new origin coordinates and the Euler angles which define the natural axes of the material (see the detailed description of the entry: **GEOMETRY POLARIZATION**).

COMMENTS.

.Generally, the aspect ratio has to be less than 3.
.The entry "**GEOMETRY**" has to be issued under the special form "**GEOMETRY POLARIZATION** (CARTESIAN, CYLINDRICAL, SPHERICAL)".

HEXA20F (R37810).

DESCRIPTION.

.**HEXA20F** is a twenty node isoparametric hexaedra, used for fluid homogeneous media.

.The degree of freedom is, for each node: **p** (the pressure).

TOPOLOGY.

```
                                   7 . . . .20 . . . . 8
                                   . .                 . .
                                 . . .               . . .
                               18 .                 19 .
                                . 15                 . 16
                               . .                  . .
                              5 . . . .17 . . . . 6    .
                              .    .                .    .
                              .    3 . . . .12 . . . . 4
                              .  .                  .  .
 Z                           13 .                 14 .
 . Y                          . 10                 . 11
 . .                          . .                  . .
 . .                          . .                  . .
 O . . . X                    1 . . . . 9 . . . . 2
```

ENTRIES.

."ELEMENTS" entry parameters.
HEXA20F MATER
N1 N2 N3 N4 N5 N6 N7 N8 N9 N10 N11 N12 N13 N14 N15 N16 N17 N18 N19 N20
where N1,...N20 are the actual node numbers corresponding to the integers 1,...20 on the above figure.

."MATERIALS" entry parameters.
COMP .0 RO
where COMP is the modulus of compression (Pa) and RO is the density (kg/m^3).

."GEOMETRY" entry parameters.
No.

COMMENTS.

.Generally, the aspect ratio has to be less than 3.

PRIS15F (R37820).

DESCRIPTION.

.PRIS15F is a fifteen node isoparametric triangular prism, used for fluid homogeneous media.

.The degree of freedom is, for each node: p (the pressure).

TOPOLOGY.

ENTRIES.

"ELEMENTS" entry parameters
PRIS15F MATER
N1 N2 N3 N4 N5 N6 N7 N8 N9 N10 N11 N12 N13 N14 N15
where N1 ... N15 are the actual node numbers corresponding to the integers 1 ... 15 on the above figure.

"MATERIALS" entry parameters
COMP 0 RO
where COMP is the modulus of compression (Pa), and RO is the density (kg/m^3).

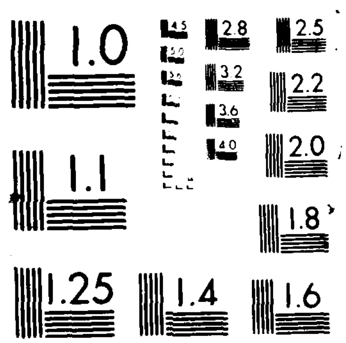
"GEOMETRY" entry parameters
No

COMMENTS.

.Generally, the aspect ratio has to be less than 3.

AD-A185 948 USER MANUAL FOR ATILA A FINITE-ELEMENT CODE FOR MODELING PIEZOELECTRIC TRANSDUCERS(U) NAVAL POSTGRADUATE SCHOOL MONTEREY CA J DECARPIGNY ET AL. SEP 87 NPS61-87-007 2/2

UNCLASSIFIED F/G 17/1 NL


1.0
2.8
2.5
3.2
2.2
3.6
4.0
2.0
1.1
1.8
1.25
1.4
1.6

AXIS08F (R37531).

DESCRIPTION.

.AXIS08F is an eight node isoparametric quadrilateral, used for fluid homogeneous media and for axisymmetrical analysis.

.The degree of freedom is, for each node: p (the pressure).

TOPOLOGY.

```
                          3 . . . . 8 . . . . 4
                          .                   .
                          .                   .
                          .                   .
                          .                   .
                          6                   7
           Y              .                   .
           .              .                   .
           .              .                   .
           .              .                   .
           0 . . . X      1 . . . . 5 . . . . 2
```

ENTRIES.

."ELEMENTS" entry parameters.
AXIS08F MATER
N1 N2 N3 N4 N5 N6 N7 N8
where N1,...N8 are the actual node numbers corresponding to the integers 1,...8 on the above figure.

."MATERIALS" entry parameters.
COMP .0 RO
where COMP is the modulus of compression (Pa) and RO is the density (kg/m^3).

."GEOMETRY" entry parameters.
No.

COMMENTS.

.Generally, the aspect ratio has to be less than 3.

AXIS06F (R37532).

DESCRIPTION.

.AXIS06F is a six node isoparametric triangle used for fluid homogeneous media and for axisymmetrical analysis.

.The degree of freedom is, for each node: p (the pressure).

TOPOLOGY.

ENTRIES.

."ELEMENTS" entry parameters.
AXIS06F MATER
N1 N2 N3 N4 N5 N6
where N1,...N6 are the actual node numbers corresponding to the integers 1,...6 on the above figure.

."MATERIALS" entry parameters.
COMP .0 RO
where COMP is the modulus of compression (Pa) and RO is the density (kg/m^3).

."GEOMETRY" entry parameters.
No.

COMMENTS.

.Generally, the aspect ratio has to be less than 3.

QUAD16I (R37910).

DESCRIPTION.

.QUAD16I is a sixteen node isoparametric quadrilateral element which has to be used to ensure the matching between solid and fluid meshes along their interface in the case of three-dimensional analyses. It includes 8 solid nodes and 8 fluid nodes which have the same coordinates as the solid nodes. It has no thickness.

.The degrees of freedom are, for each solid node: u_x, u_y, u_z (3 translations), and for each fluid node: p (the pressure) .

TOPOLOGY.

```
                          SOLID MESH

                                 7 . . . .16 . . . . 8
                               . 3 . . . .12 . . . . 4
                             . .                   . .
                           . .                   . .
                        14 .                  15 .
 Z       Y             .10                  .11
 .     .             . .                  . .
 .   .             . .                  . .
 . .             5 . . . .13 . . . . 6 .
 O . . . X       1 . . . . 9 . . . . 2

                                        FLUID MESH
```

ENTRIES.

."ELEMENTS" entry parameters.
QUAD16I
N1 N2 N3 N4 N5 N6 N7 N8 N9 N10 N11 N12 N13 N14 N15 N16
where N1,...N16 are the actual node numbers corresponding to the integers 1,...16 on the above figure.

."MATERIALS" entry parameters.
No.

."GEOMETRY" entry parameters.
No.

COMMENTS.

.The nodes 1, 2, 3, 4, 9, 10, 11, 12 are fluid nodes. The nodes 5, 6, 7, 8, 13, 14, 15, 16 are solid nodes.
.Attention must be paid to the orientation of the OZ axis, from the fluid domain to the solid domain.

TRIA121 (R37920).

DESCRIPTION.

.TRIA121 is a twelve node isoparametric triangular element which has to be used to ensure the matching between solid and fluid meshes along their interface in the case of three-dimensional analyses. It includes 6 solid nodes and 6 fluid nodes which have the same coordinates as the solid nodes. It has no thickness.

.The degrees of freedom are, for each solid node: u_x, u_y, u_z (3 translations), and for each fluid node: p (the pressure) .

TOPOLOGY.

```
                                 SOLID MESH
                                               6
                                            . .3
                                        .   . . .
                                    .   .     . .
                               12   .       . .
Z        Y                  .   9         .8
.     .                .   .             . .
.   .              .   .                . .
. .          4 . . . . .10 . . . . . 5.
0 . . . X    1 . . . . . 7 . . . . . 2

                                     FLUID MESH
```

ENTRIES.

."ELEMENTS" entry parameters.
TRIA121
N1 N2 N3 N4 N5 N6 N7 N8 N9 N10 N11 N12
where N1,...N12 are the actual node numbers corresponding to the integers 1,...12 on the above figure.

."MATERIALS" entry parameters.
No.

."GEOMETRY" entry parameters.
No.

COMMENTS.

.The nodes 1, 2, 3, 7, 8, 9 are fluid nodes. The nodes 4, 5, 6, 10, 11, 12 are solid nodes.
.Attention must be paid to the orientation of the OZ axis, from the fluid domain to the solid domain.

AXIS06I (R37541).

DESCRIPTION.

.AXIS06I is a six node isoparametric element which has to be used to ensure the matching between solid and fluid meshes along their interface in the case of axisymmetrical analyses. It includes 3 solid nodes and 3 fluid nodes which have the same coordinates as the solid nodes. It has no thickness.

.The degrees of freedom are, for each solid node: u_x, u_y (2 translations), and for each fluid node: **p** (the pressure) .

TOPOLOGY.

```
Y
.                      SOLID MESH
.
.                    3 . . . . 6 . . . . 4
O . . . X            1 . . . . 5 . . . . 2

                       FLUID MESH
```

ENTRIES.

."ELEMENTS" entry parameters.
AXIS06I
N1 N2 N3 N4 N5 N6
where N1,...N6 are the actual node numbers corresponding to the integers 1,...6 on the above figure.

."MATERIALS" entry parameters.
No.

."GEOMETRY" entry parameters.
No.

COMMENTS.

.The nodes 1, 2, 5 are fluid nodes. The nodes 2, 3, 6 are solid nodes.
.Attention must be paid to the **orientation of the OY axis**, from the fluid domain to the solid domain.

QUAD08M (R37930).

DESCRIPTION.

.**QUAD08M** is an eight node isoparametric quadrilateral damping element used to take account of a radiation condition. It is restricted to the monopolar (spherical) damping condition. It has to be spherical and attached to the outside surface of a three-dimensional fluid mesh.

.The degree of freedom is, for each node: p (the pressure).

TOPOLOGY.

```
                      3 . . . . 8 . . . . 4
                      .                   .
                      .                   .
                      .                   .
                      .                   .
                      6                   7
    Y                 .                   .
    .                 .                   .
    .                 .                   .
    .                 .                   .
    0 . . . X         1 .   . . 5 . . . . 2
```

ENTRIES.

."ELEMENTS" entry parameters.
QUAD08M MATER NG
N1 N2 N3 N4 N5 N6 N7 N8
where N1,...N8 are the actual node numbers corresponding to the integers 1,...8 on the above figure.

."MATERIALS" entry parameters.
COMP .0 RO
where COMP is the modulus of compression (Pa) and RO is the density (kg/m^3).

."GEOMETRY" entry parameter.
R
where R is the radius of curvature of the element.

COMMENTS.

.Generally, the aspect ratio has to be less than 3.
.With respect to the orientation of the whole mesh, <u>its normal has to point outwards</u>.

TRIA06M (R37940).

DESCRIPTION.

.TRIA06M is a six node isoparametric triangular damping element used to take account of a radiation condition. It is restricted to the monopolar (spherical) damping condition. It has to be spherical and attached to the outside surface of a three-dimensional fluid mesh.

.The degree of freedom is, for each node: p (the pressure).

TOPOLOGY.

ENTRIES.

."ELEMENTS" entry parameters.
TRIA06M MATER NG
N1 N2 N3 N4 N5 N6
where N1,...N6 are the actual node numbers corresponding to the integers 1,...6 on the above figure.

."MATERIALS entry parameters.
COMP .0 RO
where COMP is the modulus of compression (Pa) and RO is the density (kg/m^3).

."GEOMETRY" entry parameter.
R
where R is the radius of curvature of the element.

COMMENTS.

.Generally, the aspect ratio has to be less than 3.
.With respect to the orientation of the whole mesh, <u>its normal has to point outwards</u>.

AXIS03M (R37551).

DESCRIPTION.

.AXIS03M is a three node isoparametric damping element used to take account of a radiation condition. It is restricted to the monopolar (spherical) damping condition. It has to be circular and attached to the outside surface of an *axisymmetrical fluid mesh*.

.The degree of freedom is, for each node: p (the pressure).

TOPOLOGY.

```
Y
.
.
.
O . . . X          1 . . . . 3 . . . . 2
```

ENTRIES.

."ELEMENTS" entry parameters.
AXIS03M MATER NG
N1 N2 N3
where N1, N2 and N3 are the actual node numbers corresponding to the integers 1, 2 and 3 on the above figure.

."MATERIALS" entry parameters.
COMP .0 RO
where COMP is the modulus of compression (Pa) and RO is the density (kg/m^3).

."GEOMETRY" entry parameter.
R
where R is the radius of curvature of the element.

COMMENTS.

.To ensure the correct orientation of this element, its local OX axis must be oriented in the positive direction of the global OX axis. Thus, the global x coordinate of node 2 must be higher than the global x coordinate of node 1 on the above figure.

AXISO3D (T37551).

DESCRIPTION.

.AXISO3D is a three node isoparametric damping element used to take account of a radiation condition. It includes the dipolar damping condition. It has to be circular and attached to the outside surface of an axisymmetrical fluid mesh.

.The degree of freedom is, for each node: p (the pressure).

TOPOLOGY.

```
Y
.
.
.
O . . . X            1 . . . . 3 . . . . 2
```

ENTRIES.

."ELEMENTS"entry parameters.
AXISO3D MATER NG
N1 N2 N3
where N1, N2 and N3 are the actual node numbers corresponding to the integers 1, 2 and 3 on the above figure.

."MATERIALS" entry parameters.
COMP .0 RO
where COMP is the modulus of compression (Pa) and RO is the density (kg/m^3).

."GEOMETRY" entry parameter.
R
where R is the radius of curvature of the element.

COMMENTS.

.To ensure the correct orientation of this element, its local OX axis must be oriented in the positive direction of the global OX axis. Thus, the global x coordinate of node 2 must be higher than the global x coordinate of node 1 on the above figure.

```
 AAAAAAAA    TTTTTTTTTT   II   LL          AAAAAAAA
AA      AA       TT       II   LL         AA      AA
AA      AA       TT       II   LL         AA      AA
AAAAAAAAAA       TT       II   LL         AAAAAAAAAA
AA      AA       TT       II   LL         AA      AA
AA      AA       TT       II   LL         AA      AA
AA      AA       TT       II   LLLLLLL    AA      AA
```

FINITE ELEMENT MODELING

OF

PIEZOELECTRIC TRANSDUCERS

AUGUST 1987

CHAPTER 5
DESCRIPTION OF AN ATILA RUN

THE FINITE ELEMENT CODE ATILA HAS BEEN DESIGNED BY THE ACOUSTICS LABORATORY OF THE "INSTITUT SUPERIEUR D'ELECTRONIQUE DU NORD" (ISEN. LILLE. FRANCE) FOR THE "GROUPE D'ETUDES ET DE RECHERCHE EN DETECTION SOUS-MARINE" (GERDSM. DCAN. FRENCH NAVY. TOULON). THE SOFTWARE ENGINEERING IS PROVIDED BY THE SINAPTEC COMPANY (LILLE. FRANCE).

5.1 INTRODUCTION

The general organisation of an **ATILA** job is described in **section 1.3**. This chapter provides the corresponding organisation of an **ATILA** run in the special case of a VAX or MICROVAX computer using the VMS operating system. The same organisation is also correct for other computer types but with different procedures. Moreover, the part of this chapter which is devoted to the graphic displays implies that the **GKS library** is available and can be linked during the corresponding part of the run, the user being then working interactively with the help of a graphic terminal (Tektronix 4014, VT240 ...).

5.2 THE MAIN RUN

To run an **ATILA** job, the user must have written a data file as described in **chapter 3**. This data file must be named **JOBNAME.ATI**, ATI being the job type. Then the user must run the automatic program generator, named **PGEN**, using the entry:

RUN PGEN

During this run, several questions are asked by the code which allow the specification of the job name (JOBNAME) and the job type (ATI), the verification of the analysis type which has been identified by the code, the requirement of special data storages for a job restart (only in the case of an harmonic analysis) or a postprocessing (graphic displays ...). **PGEN** provides three files named **JOBNAME.FOR**, **JOBNAME.COM** and **JOBNAME.LST**. JOBNAME.FOR contains the **ATILA** main program. Generally, it can be used without any modification. Nevertheless, it has to be edited in few special cases: the modal analysis of an elastic structure flooded in an incompressible fluid (MOD 5), the radiated pressure computation for a given displacement field (HAR 3) and the use, by advanced users, of new algorithms which are currently under test (MOD 1, MOD 4, ...). In these cases, footnotes are available at the end of the corresponding sections in **chapter 2** which provide all the necessary information. JOBNAME.COM is used to compile and link the program. It need not be edited. Finally, JOBNAME.LST contains a copy of the data file, as read during the **PGEN** run, as well as information about the degree of freedom numbering. It allows a very efficient check of the data file and mainly of the boundary conditions. Generally, the user must edit this list and perform the corresponding check.

After the **PGEN** run, the user has to compile the main progam and link it to the requested libraries. This step is simply realised using the entry:

@JOBNAME

which provides the executable (**JOBNAME.EXE**). Finally, the user has to run the program using the entry:

RUN JOBNAME

During this run, the code asks for the job name (JOBNAME) and the job type (ATI) and information appears on the user's screen which describes the current status (number of the element the code is currently assembling, name of the solver the code is currently using ...) and provides the timing. Moreover, the code writes down several files named **JOBNAME.LST**, **JOBNAME.WRK**, **JOBNAME.SY2** and

JOBNAME.SY4. JOBNAME.LST contains a partial copy of the data file, depending upon the printing level selected by the user in this data file, and all the results (eigenfrequencies, displacement or pressure fields associated with successive eigenmodes or excitation frequencies, electrical impedances ...). The JOBNAME.WRK and JOBNAME.SY2 files are working arrays the code always needs. They contain useful information at the end of the run only if a restart has to be performed. The JOBNAME.SY4 file contains several arrays the code needs for postprocessing. Particularly, it can provide the PLO array which contains the successive displacement and/or pressure fields as well as the CPDDC array which contains the node coordinates and the degree of freedom numbering. This file can be easily read by advanced users, by editing the JOBNAME.FOR file, deleting all the subroutine calls and replacing the WRITE entries devoted to the SY4 file by the corresponding READ entries.

5.3 THE RESTART

To restart an ATILA job is often necessary when the user is performing an harmonic analysis, since more than 10 or 20 different frequencies are often required in the frequency range of interest. The restart allows the computation of the frequency response using previously assembled matrices and often saves a large amount of CPU time. Obviously, it can only be performed if a full run has been realised before and if the required arrays have been stored. Then, the process is exactly the same as for a main run, except that the user has to select the restart during the run of PGEN, when the corresponding question appears on the screen.

5.4 PRE AND POSTPROCESSING

In this version of the **ATILA** code, the available pre and postprocessors are mainly devoted to the graphic display using the **GKS** package, although many specific postprocessors (farfield computation, transmitting voltage response and directivity computations and displays, stress computations and displays, isovalue displays ...) are currently under test but are not documented for the present time. At the preprocessing level, the code can provide graphic displays of the mesh with different types of numbering (nodes, elements, degrees of freedom). At the postprocessing level, it can provide the same graphic displays as well as distorted shapes of the structure which describe the resulting displacement fields. Detailed information about these graphic displays are given in **section 3.6**. To obtain a graphic display, the user must make the entry:

RUN PGRAPH

Then, **PGRAPH** reads the JOBNAME.ATI file and writes down three files named **DJOBNAME.LST**, **DJOBNAME.FOR** and **DJOBNAME.COM**. Moreover, during the run, questions appear on the screen which are very similar to the question which appears during a **PGEN** run. The DJOBNAME.LST file is identical to the JOBNAME.LST file which is obtained after a **PGEN** run and allows a detailed check of the data file if **PGRAPH** is run at the preprocessing level. The DJOBNAME.FOR file contains the main program which is needed to obtain the graphic displays. It need not be edited. Finally, the DJOBNAME.COM file is used to compile and link the program*. Generally, it need not be edited.

After the **PGRAPH** run, the user must compile the main program and link it to the requested libraries. This step is realised simply using the entry:

@DJOBNAME

which provides the executable (**DJOBNAME.EXE**). Finally, the user must run the program using the entry:

RUN DJOBNAME

During this run, the code asks the job name (JOBNAME), the job type (ATI) and requires the choice of the pre or postprocessing level. The preprocessing level can be used as soon as the JOBNAME.ATI file is available. The postprocessing level can only be used if a full **ATILA** run has previously been performed (main run or restart) and if the necessary storages have been made. The whole graphic display phase must be performed using a graphic terminal and the information about the display type, its orientation, its title ..., are required during the run by questions which are written down on the screen.

*. The link of the main program to the GKS library is provided by the file LNK1D.COM which is used by DJOBNAME.COM. Before the first PGRAPH run, the user must check that this link is correct, taking account of the GKS installation on the corresponding computer.

5.5 THE AUTOMATIC MESH GENERATION

The use of the automatic mesh generator **MOSAIQUE** is described in the following chapter. It requires a **JOBNAME.MOS** file which is written down by the user. Then the user runs **MOSAIQUE** with the entry:

RUN MOSAIQUE

MOSAIQUE provides two files named **JOBNAME.LST** and **JOBNAME.ATI**. JOBNAME.LST contains information which are only needed if a problem appears during the generation. The JOBNAME.ATI file is the data file for the **ATILA** main run. Depending upon the contents of the JOBNAME.MOS file, it can be used immediately or can be edited by the user to add the last entries and data, generally the loading or boundary conditions.

5.6 SUMMARY

Automatic mesh generation.

JOBNAME.MOS

RUN MOSAIQUE

JOBNAME.LST
JOBNAME.ATI

Preprocessing.

JOBNAME.ATI

RUN PGRAPH

DJOBNAME.LST
DJOBNAME.FOR
DJOBNAME.COM

@DJOBNAME

DJOBNAME.EXE

RUN DJOBNAME

Interactive graphic display

Main run.

JOBNAME.ATI

RUN PGEN

JOBNAME.LST
JOBNAME.FOR
JOBNAME.COM

@JOBNAME

JOBNAME.EXE

RUN JOBNAME

JOBNAME.LST
JOBNAME.WRK
JOBNAME.SY2
JOBNAME.SY4

Restart.

```
JOBNAME.ATI
JOBNAME.WRK
JOBNAME.SY2
JOBNAME.SY4

RUN PGEN

JOBNAME.LST
JOBNAME.FOR
JOBNAME.COM

@JOBNAME

JOBNAME.EXE

RUN JOBNAME

JOBNAME.LST
JOBNAME.WRK
JOBNAME.SY2
JOBNAME.SY4
```

Postprocessing.

```
JOBNAME.ATI
JOBNAME.SY4

RUN PGRAPH

DJOBNAME.LST
DJOBNAME.FOR
DJOBNAME.COM

@DJOBNAME

DJOBNAME.EXE

RUN DJOBNAME

Interactive graphic display
```

Due to their large size, the files .LST, .WRK, .SY2, .SY4 are not stored as successive versions if they are provided several times with the same job name but are automatically deleted by the code. If, for any reason, the user wishes to save one of the previous version before a new run with the same job name, it has to rename the corresponding file.

```
 AAAAAAAA    TTTTTTTTTT   II  LL          AAAAAAAA
AA      AA       TT       II  LL         AA      AA
AA      AA       TT       II  LL         AA      AA
AAAAAAAAAA       TT       II  LL         AAAAAAAAAA
AA      AA       TT       II  LL         AA      AA
AA      AA       TT       II  LL         AA      AA
AA      AA       TT       II  LLLLLLL    AA      AA
```

FINITE ELEMENT MODELING OF PIEZOELECTRIC TRANSDUCERS

AUGUST 1987

CHAPTER 6
AUTOMATIC MESH GENERATION

THE FINITE ELEMENT CODE **ATILA** HAS BEEN DESIGNED BY THE ACOUSTICS LABORATORY OF THE **"INSTITUT SUPERIEUR D'ELECTRONIQUE DU NORD"** (ISEN. LILLE. FRANCE) FOR THE **"GROUPE D'ETUDES ET DE RECHERCHE EN DETECTION SOUS-MARINE"** (GERDSM. DCAN. FRENCH NAVY. TOULON). THE SOFTWARE ENGINEERING IS PROVIDED BY THE **SINAPTEC** COMPANY (LILLE. FRANCE).

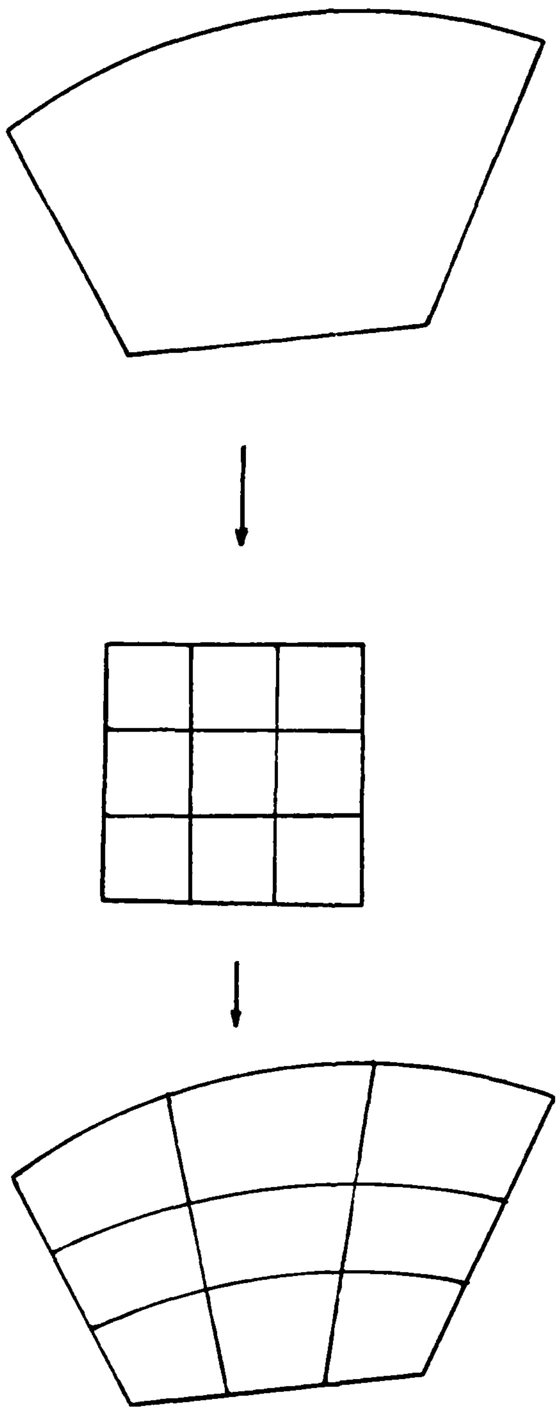
6.1 INTRODUCTION

As mentioned in **section 1.3.2** and in the introduction of the **third chapter**, partial regularity of the mesh can often allow the use of an automatic generation, which provides node coordinates and element topologies. In the **ATILA** code, this work is performed by a specialized preprocessor named **MOSAIQUE**. Then, the user has only to define a gross splitting, built with **super elements**, and to select their automatic splitting into finite elements (Figure 1). From the numerical point of view, each super element is transformed into its natural axes, using the isoparametric transformation. Then, it is split into finite elements, following the corresponding data provided by the user, and these finite elements are transformed back to the global axes using the reverse isoparametric transformation. Finally, redundant nodes are eliminated by searching nodes which belong to a same given small 2D or 3D window.

Entries for **MOSAIQUE** are provided by a specific input file written by the user and named JOBNAME.MOS. At the end of the **MOSAIQUE** run, the output file which contains the generated node coordinates and topologies is named JOBNAME.ATI (see **chapter 5**). Moreover, entries and data which are required by **ATILA** but are not necessary for the mesh generation can be also read by **MOSAIQUE**, if present in the JOBNAME.MOS file, and be copied in the JOBNAME.ATI output file. Thus, starting with a simple input file (JOBNAME.MOS), **MOSAIQUE** is able to provide the whole **ATILA** data file (JOBNAME.ATI). Finally, it has to be noted that, because the number of nodes which have to be defined for **MOSAIQUE** is often small, this procedure allows the user to modify very easily geometrical parameters (length, width ...) as well as the mesh steps.

Most of the entries and data contained in the **MOSAIQUE** data file are identical to the **ATILA** entries and data. However, the **NODES** and **ELEMENTS** entries are modified, while two new entries can be used. These entries and data are described in section 6.2. Moreover, the splitting of a super element into finite elements can be obtained using different frames. The frames which are retained in **MOSAIQUE** are described in section 6.3.

FIGURE 1. SCHEMATIC DESCRIPTION OF THE GENERATION PROCESS.



6.2 MOSAIQUE ENTRIES AND DATA

6.2.1. THE HEADER.

Information contained in section 3.2 is correct for the header of a **MOSAIQUE** data file. This header is reproduced as the header of the resulting JOBNAME.ATI file. Thus, it will be the header for the **ATILA** job.

6.2.2. THE ENTRIES.

The **ATILA** entries **ANALYSIS**, **CLASS**, **NLOAD**, **PRECISION**, **LCPDDC**, **MATERIALS**, **GEOMETRY**, **GEOMETRY POLARIZATION**, **REDUCTIONS**, **FREQUENCIES**, **RADIATION** and **PRINTING**, if provided in the **MOSAIQUE** data file, are simply read and copied in the resulting JOBNAME.ATI file. Obviously, they are not used by the mesh generation procedure. The **END** entry has to be provided to **MOSAIQUE**. As well as for **ATILA**, it defines the end of the entry list. It will also be copied in the resulting JOBNAME.ATI file.

The **NODES** entry is used to provide the coordinates of the nodes which define the super elements. Its syntax is exactly the same as for the **ATILA** *data file and the node coordinates have to be listed* in the node number ascending order associated to the gross mesh. After the **MOSAIQUE** run, this **NODES** entry will be copied in the JOBNAME.ATI file with the coordinate list which is associated with the *generated nodes. The new node numbering* will be the node numbering provided by this list. It has to be noted that the numbering of the nodes belonging to the initial gross mesh is modified during the generation. Moreover, as well as for **ATILA**, nodes have to be set in the compulsory order: solid, fluid, radiating. Finally, the entry **NEWAXES** and **SCALE** can be used to define the node coordinates for the gross mesh as well as they are used for **ATILA**.

As described in **chapter 3**, the **ELEMENTS** entry is divided in sets, each set being related to a given element type*. Successive sets have to be separated by a dummy line and the whole entry must also be closed by a dummy line. However, with respect to **ATILA**, the content of a set is modified in **MOSAIQUE** by replacing each topology line of the type:

N11 N12 N13 . . . . N1P

by a set of *lines of the type:*

N11 N12 N13 . . . . N1P
-1 ITYP NK NL NM NN I
IX1 IX2 IX3 . . . . IXNK
IY1 IY2 IY3 . . . IYNL
IZ1 IZ2 IZ3 . . . . . . IZNM

where:

- **N11, N12, N13, ..., N1P** are the actual numbers of the corresponding super element nodes,
- the second line (-1 **ITYP NK NL NM NN I**) provides compulsory information about the splitting of this super element,
- **ITYP** refers to a given frame (see section 6.3),
- **NK** is the number of generated elements along the OX natural axis of the super element,
- **NL** is the number of generated elements along the OY natural axis of the super element (if necessary),
- **NM** is the number of generated elements along the OZ natural axis of the super element (if necessary),
- **NN** is a dummy field, kept for special uses,
- **I** is a control integer which must be equal to 0 if the splittings are regular and 1 if at least one splitting is irregular,
- **IX1, IX2, IX3, ..., IXNK** define the splitting along the OX natural axis of the super element.
- **IY1, IY2, IY3, ..., IYNL and IZ1, IZ2, IZ3, ..., IZNM** provide the same information, respectively for the OY and OZ natural axes of the super element.

Then, along the OX natural axis of the super element, the corresponding side is divided into NK different parts, the lengths of which are respectively equal to:

IX1.L/IX, IX2.L/IX, IX3.L/IX, ..., IXNK.L/IX

where L is the total length, IX is given by:

IX = IX1 + IX2 + IX3 + ... + IXNK

and IX1, IX2, IX3, ...,IXNK have to be integers. The same splitting rule is applied for the OY and OZ natural directions. If I **is equal to 0**, the three lines which are devoted to the description of the splitting are not needed. If I **is equal to 1**, these three lines must be provided, even if one or two of them correspond to a regular splitting.
In the super element topology, a **mid side node** can always be omitted if the corresponding side is straight. Nevertheless, if a topology line is incomplete, it has to be completed with zeros, at least until the last defined node. As an example, if the mid side nodes 12, 14 and 21 can be omitted in the topology line **1 3 5 9 12 14 16 21**, this line can be written:

1 2 5 9 0 0 16 0 or **1 2 5 9 0 0 16**

In opposition to the **ATILA** case, when solid and fluid elements are used together, the elements must be provided in **the order: solid, interface, fluid, damping**. This order can be modified by the user in the JOBNAME.ATI file after the **MOSAIQUE** run, if needed.

Finally, two new entries are available for **MOSAIQUE** with respect to **ATILA**. The first is used to provide information related to the generation of the damping elements and must be written in the following form:

SORTFLUID
NFLU NAC NSH

where **NFLU** is the actual number of the first node belonging to the boundary surface of the gross mesh, **NAC** is the actual number of the node which is the boundary surface centre (acoustic centre) and **NSH** is a dummy field kept for special uses. If the boundary surface centre does not belong to the mesh and cannot be identified with a node, **NCA** has to be equal to zero and the origin of the global axes must be this centre. This entry is **compulsory** if damping elements are used. The second new entry is used to modify the size of the 2D or 3D window which allows the elimination of redundant nodes. It has to be used if redundant nodes are still present in the JOBNAME.ATI file or if actual nodes have been deleted. It has to be written under the form:

MERGE
XM

where **XM** will be the new size of the window (**XM** is a length and must be provided in meters). This entry has only to be used if the above mentioned problems appear, which could be the case if the mesh is very irregular. Then the best way of chosing the **XM** value is a trial and error method.

In fact, the element type is related simultaneously to the super element and the generated finite elements. This situation is simple when the super element and the generated elements correspond to the same geometrical shape (splitting of a quadrilateral super element into QUAD08E finite elements, splitting of a triangular super element into AXIS06P finite elements ...) but can lead to severe difficulties if geometrical shapes are different (splitting of a quadrilateral super element into TRIA06E finite elements, as an example). In this last case, the user must provide an element type which corresponds to the geometrical shape of the super element. Then, **MOSAIQUE** writes down this wrong element type in the JOBNAME.ATI file and the user has to modify this type before using the data file. Work is in progress to avoid this problem in the next **ATILA** version.

6.2.3. THE DATA.

Data can be simply added to the entries in the JOBNAME.ATI file, after the automatic mesh generation. This procedure is classical, since the node numbers which are necessary to write down the data are only available after the generation. However, in special cases (multiple runs with the same topology and different dimensions, for example), the user may wish to write down these data in the JOBNAME.MOS file. Then, the rules are the same as for an **ATILA** data file and the corresponding lines are simply read by **MOSAIQUE** and copied into the JOBNAME.ATI file.

6.3 THE GENERATING FRAMES

The splitting of a super element in simple finite elements can be realised in various ways, depending upon the shape of this super element and the type of the requested finite elements (quadrilateral 2D elements with 8 nodes, 20 node hexaedras, 6 node axisymmetrical interfaces ...). The available types of splitting, named **frames** in the previous sections, are marked by the ITYP control integer and described in the following lines.

ITYP = 1

A linear super element is split into 2 node linear finite elements (SPRI02E) following the diagram:

1 . . . 0 . . . 0 . . . 0 . . . 2

NK elements (NK = 2)

The super element is defined by nodes 1 and 2 and the control integers NL and NM are dummy.

ITYP = 2

A linear super element is split into 3 node damping elements (AXIS03M or AXIS03D) following the diagram:

1 . . . 0 . . . 0 . . . 0 . . . 2

NK elements (NK = 2)

The super element is defined at least by the nodes 1 and 2 but generally also by its mid node, due to the curved shape of the damping boundary. The control integers NL and NM are dummy.

ITYP = 3

A linear super element is split into 6 node interface elements (AXIS06I) following the diagram:

```
                Fluid domain

    0 . . . 0 . . . 0 . . . 0 . . . 0
    1 . . . 0 . . . 0 . . . 0 . . . 2

                Solid domain

            NK elements (NK = 2)
```

The super element is defined by the nodes 1 and 2. Its mid node is required only if the interface is curved. The control integers NL and NM are dummy. The orientation of the normal to the interface from the fluid to the solid being compulsory for the generated interface finite elements, the orientation of the super element is also compulsory. **Nodes 1 and 2 are defined in the above order if the Oz natural axis is the outward normal to this paper sheet.** Then, the Ox natural axis is oriented toward the left (attention must be devoted to this point, since this numbering rule which applies to the super element is the opposite of the rule which applies to the finite elements).

ITYP = 4

A quadrilateral super element is split into 8 node quadrilateral elements (QUAD08E, PLAT08E, FACE08E, AXIS08P, QUAD08P, AXIS08F or QUAD08M) following the diagram:

```
              3 . . 0 . . 0 . . 0 . . 0 . . 0 . . 4
              .           .           .           .
              .           .           .           .
              0           0           0           0
              .           .           .           .
     NL       .           .           .           .
  elements    0 . . 0 . . 0 . . 0 . . 0 . . 0 . . 0
  (NL = 2)    .           .           .           .
              .           .           .           .
              0           0           0           0
              .           .           .           .
              .           .           .           .
              1 . . 0 . . 0 . . 0 . . 0 . . 0 . . 2

                      NK elements (NK = 3)
```

The super element is defined by the nodes 1, 2, 3 and 4. Mid side nodes are only required if the sides are curved. The control integer NM is dummy.

ITYP = 5

A quadrilateral super element is split into 6 node triangular elements (TRIA06E, PLAT06E, AXIS06P, TRIA06P, AXIS06F or TRIA06M) following the diagram:

```
                 3 . . . 0 . . . 0 . . . 0 . . . 4
                 . .           . .           . .
                 .   .       .     .       .   .
                 0     0     0     0     0     0
                 .       .   .       .   .       .
      NL         .         . .         . .         .
   elements      0 . . 0 . . 0 . . 0 . . 0 . . 0 . . 0
   (NL = 3)      .         . .         . .         .
                 .       .   .       .   .       .
                 0     0     0     0     0     0
                 .   .       .     .       .   .
                 . .           . .           . .
                 0 . . . 0 . . . 0 . . . 0 . . . 0
                 . .           . .           . .
                 .   .       .     .       .   .
                 0     0     0     0     0     0
                 .       .   .       .   .       .
                 .         . .         . .         .
                 1 . . 0 . . 0 . . 0 . . 0 . . 0 . . 2

                       NK elements (NK = 3)
```

The super element is defined by the nodes 1, 2, 3 and 4. Mid side nodes are only required if the sides are curved. The control integer NM is dummy. This case is related to the comments at the end of section 6.2.2 and care must be paid to the element type (as an example, the element type can be QUAD08E in the MOSAIQUE data file and TRIA06E in the ATILA data file).

ITYP = 6

A triangular super element is split into 6 node triangular elements (TRIA06E, PLAT06E, AXIS06P, TRIA06P, AXIS06F or TRIA06M) following the diagram:

```
                            3
                           . .
                         .     .
                        0       0
                      .           .
                    .               .
                   0  .  .  0  .  .  0
                 . .               . .
     NL        .     .           .     .
  elements    0       0         0       0
  (NL = 3)   .           .   .           .
           .               . .             .
          0  .  .  0  .  .  0  .  .  0  .  .  0
        . .                . .                . .
      .     .            .     .            .     .
     0       0          0       0          0       0
    .           .     .           .     .           .
  .               . .               . .               .
 1  .  .  0  .  .  0  .  .  0  .  .  0  .  .  0  .  .  2

                NK elements (NK = 3)
```

The super element is defined by the nodes 1, 2 and 3. Mid side nodes are only required if the sides are curved. The control integer NM is dummy.

ITYP = 9

A quadrilateral super element is split into 16 node quadrilateral interface elements (QUAD16I) following the diagram:

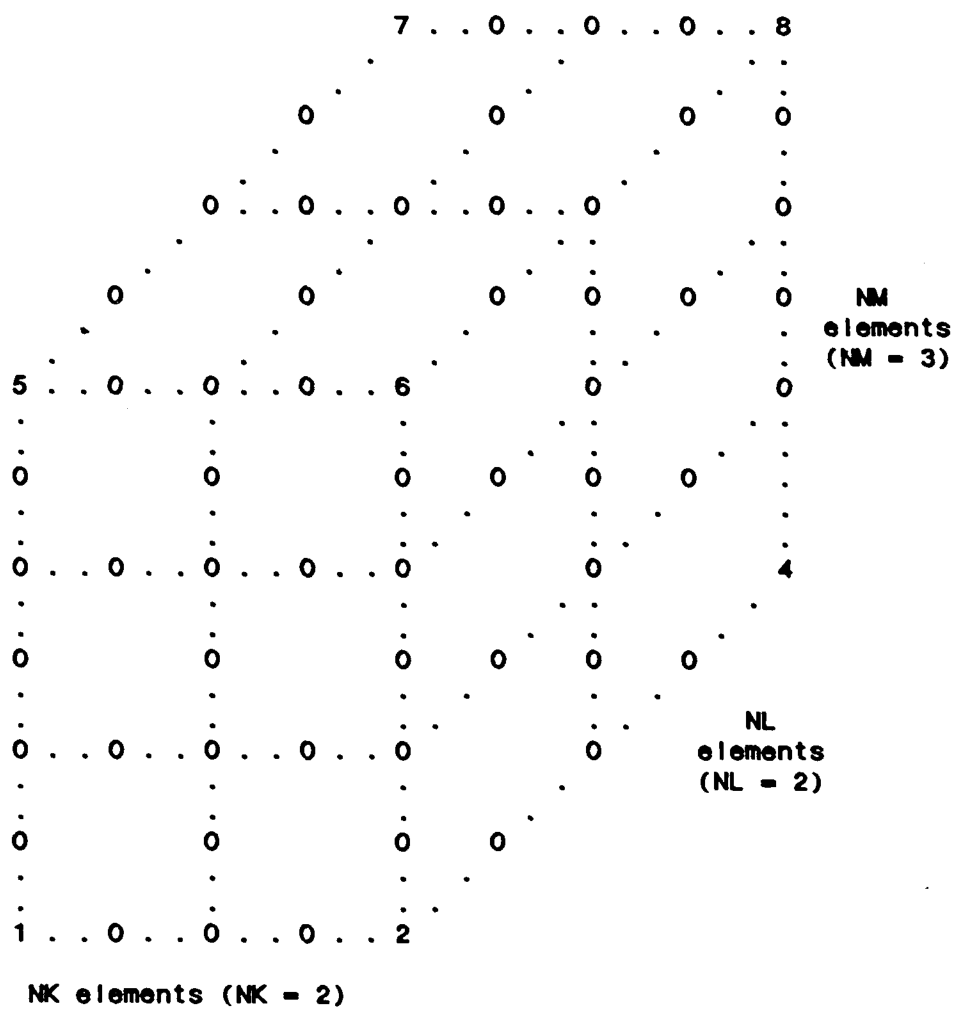
```
                              0 . . 0 . . 0 . . 0 . . 0
                             .3 . . 0 . ..0 . . 0 . ..4
                          ..          ..          ..
                       0.          0.          0.
                      .0          .0          .0
                    ..          ..          ..
   NL             0.. . 0 . . 0.. . 0 . . 0.
elements         .0 . . 0 . ..0 . . 0 . ..0
(NL = 2)       ..        0.          ..
             0.         .0          0.
            .0        ..           .0
          ..        ..          ..
        0.. . 0 . . 0.. . 0 . . 0.
        1 . . 0 . . 0 . . 0 . . 2

           NK elements (NK = 2)
```

The super element is defined by the nodes 1, 2, 3 and 4. Mid side nodes are only required if the sides are curved. The control integer NM is dummy. Attention must be paid to the orientation of this super element, the normal to the interface being always oriented from the fluid toward the solid.

ITYP = 10

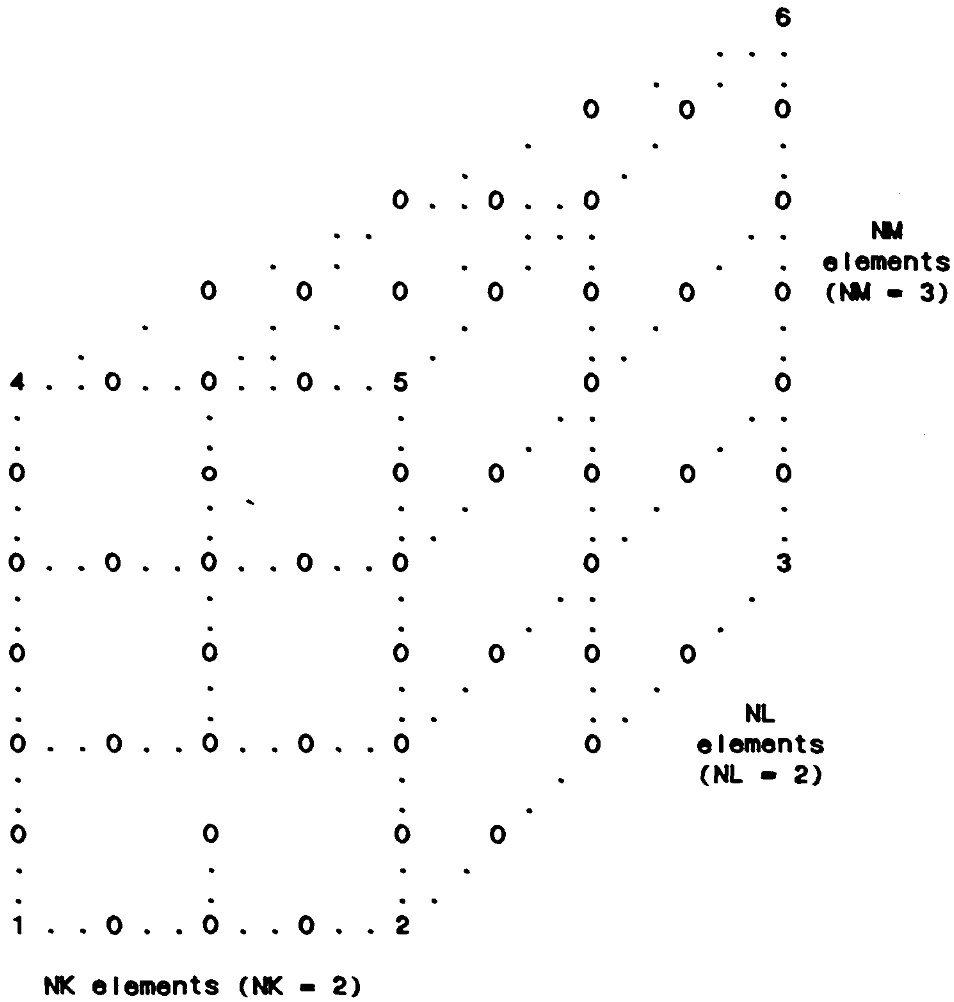
An hexaedric super element is split into 20 node hexaedric finite elements (HEXA20E, HEXA20P or HEXA20F) following the diagram:



The super element is defined by the nodes 1, 2, 3, 4, 5, 6, 7 and 8. Mid side nodes are only required if the sides are curved.

ITYP = 11

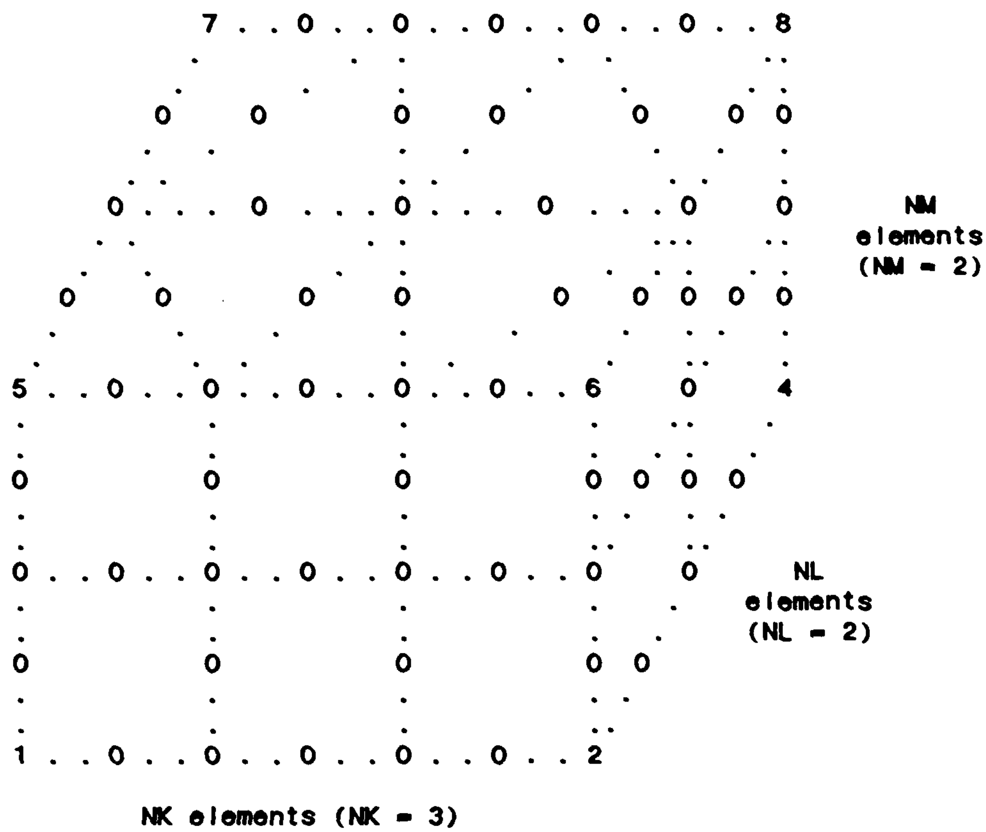
A prismatic super element with a triangular basis is split into 15 nodes prismatic finite elements (PRIS15E, PRIS15P or PRIS15F) following the diagram:



The super element is defined by nodes 1, 2, 3, 4, 5 and 6. Mid side nodes are only required if the sides are curved.

ITYP = 12

An hexaedric super element is split into 15 node prismatic finite elements (PRIS15E, PRIS15P or PRIS15F) following the diagram:



The super element is defined by the nodes 1, 2, 3, 4, 5, 6, 7 and 8. Mid side nodes are only required if the sides are curved. This case is related to the comments at the end of section 6.2.2 and care must be paid to the element type (as an example, the element type can be HEXA20E in the MOSAIQUE data file and PRIA15E in the ATILA data file).

ITYP = 14

A linear super element is split into 3 node axisymmetrical shell finite elements (SHEL03E). The information related to ITYP = 2 is also correct for this case but the numbering of the generated nodes and the topology of the generated elements take account of the specific numbering given for this shell in section 4.3

ATILA Workshop

Naval Postgraduate School
3-6 Aug 87

Participants

Steve Baker	Physics Dept. Code 61Ba Naval Postgraduate School Monterey, CA 93943 408/646-2732
George Benthien	Naval Ocean Systems Ctr Code 712 San Diego, CA 92152 619/225-6301
John Blottman	Naval Underwater Systems Center New London, CT 06320 203/440-5472
Jean-Claude Debus	Physics Dept, NPS and Institut Superieur d'Electronique du Nord 41, Blvd. Vauban 59046 Lille, Cedex, France
Jean-Noel Decarpigny	Physics Dept, NPS and Institut Superieur d'Electronique du Nord 41, Blvd. Vauban 59046 Lille, Cedex, France
Tom Gabrielson	Naval Air Development Center Code 5031 Warminster, PA 18974 215/441-1310
George Knott	Physics Department Naval Postgraduate School Monterey, CA 93943 408/646-3227
Ender Kuntsal	Physics Department Naval Postgraduate School Monterey, CA 93943 408/646-3227
Robert Montgomery	Naval Research Lab USRD P. O. Box 8337 Orlando, FL 32856 305/857-5126
Roger Richards	Naval Underwater Systems Center Code 2132 New London, CT 06320 203/440-5472

Tina Ruggiero	Naval Research Lab USRD P. O. Box 8337 Orlando, FL 32856 305/857-5182
Ken Webman	Naval Underwater Systems Center New London, CT 06320 203/440-4614
O. B. Wilson	Physics Department Naval Postgraduate School Monterey, CA 93943 408/646-2894

INITIAL DISTRIBUTION LIST

	Number of copies
Defense Technical Information Center Cameron Station Alexandria, VA 22304-6145	2
institut superieur d'electronique du nord 41 Blvd Vauban 59046 Lille, Cedex, France	
Jean-Noel Decarpigny	2
Jean-Claude Debus	1
Bernard Hamonic	1
Naval Air Development Center ATT Code 5031 Tom Gabrielson Warminster, PA 18974	1
Naval Ocean Systems Center San Diego, CA 92152	
George Benthien Code 712	2
Don Barach	1
Ed Rynne	1
Dave Gillette	1
Harry Schenck	1
Naval Postgraduate School Monterey, CA 93943	
Research Administration Code 012	1
Chairman, Department of Physics Code 61	1
Steven R. Baker Code 61BAa	1
Ender Kuntsal Code 61Ku	1
Gilles Cantin Code 69Ci	1
George Knott Code 61Kt	1
O. B. Wilson Code 61Wl	5
Library Code 0142	2
Naval Research Laboratory, USRD PO Box 8337 Orlando, FL 32856-8337	
Robert Timme	1
C. Ruggiero	2
Robert Montgomery	1
Theodore Henriquez	1
Naval Sea Systems Command ATT PMS 390 E 58 (C. Allen) Washington, DC 20362-5101	1
Naval Underwater Systems Center New London, CT 06320	
Ken Webman	2
Roger Richards Code 2132	1
John Blottman	1
Bernie McTaggert	1

END

12-87

DTIC